

LS-PRE/POST v1.0



Keyboard and Mouse Operation	13
Dynamic Model Operation	13
Drawing Area Selection	13
Lists and Multiple Selections	13
Drop Down Menus	14
File	14
Open	14
Print dialog	15
Quick Guides	15
Quick Guide: Printing drawing area to a file	15
Quick Guide: Printing drawing area to paper	15
Movie Dialog	16
Quick Guides	16
Quick Guide: How to create a movie	16
Misc.	16
Toggle	17
Background	17
Help	17
Command File (Cfile) Dialog	18
Introduction	18
Dialog Description	18
Quick Guides	18
Quick Guide: How to record a command file	18
Quick Guide: How to replay a command file	18
Ruler	19
Introduction	19
Dialog Description	19
Quick Guides	19
Quick Guide: How to use the ruler	19
Assign Menu Buttons	20
Introduction	20
Dialog Description	20
Quick Guides	20
Quick Guide: How to Assign menu buttons	20
Quick Guide: How to clear menu button assignments	20
Command Interface	21
Animation Control	22
Rendering Hot Buttons	23
Main Menu	25
Page 1:Post Processing	26
Introduction	26



Follow Options	27
Introduction	27
Interface Description	27
Quick Guides	27
Quick Guide: How to follow a node	27
Quick Guide: How to follow a plane	27
Formatted Output	28
Introduction	28
Interface Description	28
Quick Guides	28
Quick Guide: How to write an output data file	28
Annotation	29
Introduction	29
Interface Description	29
Quick Guide: How to add a label with an arrow	30
Section Plane	31
Introduction	31
Interface Description	31
Quick Guides	31
Quick Guide: How to define a plane using 1p+NL	31
Quick Guides	32
Quick Guide: How to define a plane using N1-N2	32
Quick Guide: How to define a plane using 3Nds	32
Quick Guide: How to define a plane using 2Nds+D	32
Force: Section force plot interface	33
Quick Guides	33
Quick Guide: How to display a force plot	33
MVP: Moving Section Plane Interface	34
Quick Guides	34
Quick Guide: How to setup a moving plane	34
Quick Guide: How to store and view multiple section cuts	34
Options: Section Plane Options	35
File Writing	35
Quick Guides	35
Quick Guide: How to output section cut data	35
Crush: Intrusion Plotting	36
Quick Guides	36
Quick Guide: How to display an intrusion plot	36
Line: Cut Line Plot interface	37
Quick Guides	37
Quick Guide: How to plot Cut Line data	37
Set Fringe Range	38
Introduction	38
Interface Description	38
Fringe Color Palette	39
Quick Guides	39
Quick Guide: How to set a fringe range	39
Quick Guide: How to set the fringe color palette	39
Quick Guide: How to use Show	39
Quick Guide: How to show active minima and maxima	39
Find Ele/Node/Mat	40
Introduction	40



Interface Description	40
Quick Guides	40
Quick Guide: How to find an entity	40
Fringe Component	41
Introduction	41
Interface Description	41
Quick Guides	41
Quick Guide: How to display fringe components	42
Set Part Appearance	43
Introduction	43
Interface Description	43
Quick Guides	43
Quick Guide: How to set part appearance	43
Group Operations	44
Introduction	44
Interface Description	44
Quick Guides	44
Quick Guide: How to create a group	44
Quick Guide: Available group operations	44
Split Window	45
Introduction	45
Interface Description	45
Quick Guides	45
Quick Guide: How to use multiple windows	45
Node Trace	46
Introduction	46
Interface Description	46
Quick Guides	46
Quick Guide: How to generate a node trace	46
Quick Guide: How to write a node trace curve	46
Quick Guide: How to write a node trace in text format	46
Lighting Setup	47
Introduction	47
Interface Description	47
Quick Guides	47
Quick Guide: How to setup model lighting	47
Set Display Options	48
Introduction	48
Interface Description	48
Displacement Scale Factor	48
Reflections	49
Head Injury Criteria (Hic) / Chest Severity Index (Csi)	49
SPH nodes	50
Shell Thickness Scale Factor	50
Fringe Scale Factor	50
Vector Plot	50
Introduction	51
Interface Description	51
Quick Guides	51
Quick Guide: How to display a vector plot	51



Identify Node/Element/Part	52
Introduction	52
Interface Description	52
Quick Guides	52
Quick Guide: How to identify nodes/elements/parts	52
Time History Results	53
Introduction	53
Interface Description	53
Vol-Fail: Volume failure	54
Quick Guides	54
Quick Guide: How to display a general time history plot	54
Quick Guide: How to display an element time history plot	54
Quick Guide: Using Volume Failure plotting with 1 level and n materials	54
Quick Guide: Using Volume Failure plotting with n levels and 1 material	54
Set Part Color	55
Introduction	55
Interface Description	55
Color Options	56
Transparency Options	56
Quick Guides	57
Quick Guide: How to assign a color to a part/option	57
Quick Guide: How to assign a color to the color map	57
Quick Guide: How to assign a transparency to a part	57
Element Blanking	58
Introduction	58
Interface Description	58
Quick Guides	58
Quick Guide: How to blank elements by using an area/polygon	58
Quick Guide: How to blank elements using a sphere	58
Explode Parts	59
Introduction	59
Interface Description	59
Quick Guides	59
Quick Guide: How to explode a set of parts	59
Quick Guide: Using Move	59
Cross Plotting	60
Introduction	60
Interface Description	60
Quick Guides	60
Quick Guide: Cross Plotting – File	60
Quick Guide: Cross Plotting – Window	60
XY-Plot Graphics Windows	61
Left Menu	61
Right Menu	62
Title	62
Scale	62
Attributes	63
Quick Guides	63
Quick Guide: Setting XY-Graph attributes using Auto Symbol	63
Quick Guide: Setting XY-Graph attributes manually	63
Filter	64



Quick Guides	64
Quick Guide: How to apply a filter	64
Print	64
Save	65
Quick Guides	65
Quick Guide: Saving XY-Plot data to a file	65
Load	65
Quick Guides	66
Quick Guide: How to load a XY-Plot data file	66
Operations	67
Quick Guides	67
Quick Guide: Operating on a curve	67
Quick Guide: How to subtract/multiply/divide curves	67
Quick Guide: How to apply resultant2 and 3	67
Quick Guide: How to apply Hic15/Hic36/CSI	67
Quick Guide: Saving results	67
Forming Limit Results	69
Introduction	69
Interface Description	69
Formability Diagram	70
Quick Guides	70
Quick Guide: Displaying Forming Limit Diagram	70
State Time	71
Introduction	71
Interface Description	71
Quick Guides	71
Quick Guide: How to use overlays	71
Measure	72
Introduction	72
Interface Description	72
Quick Guides	72
Quick Guide: How to measure a coordinate	72
Quick Guide: How to measure a distance	72
Quick Guide: How to measure an angle	73
Quick Guide: How to measure an area	73
Quick Guide: How to measure a volume/mass	73
Quick Guide: How to measure inertia/angular velocity	73
Quick Guide: How to define a local coordinate system	73
ASCII	74
Introduction	74
Interface Description	74
Head Injury Criteria (Hic) / Chest Severity Index (Csi) Options	75
Quick Guides	75
Quick Guide: How to load an ASCII item	75
Quick Guide: How to plot stored data	75
Quick Guide: How to set Hic/Csi options	75
Save/Retrieve Views	76
Introduction	76
Interface Description	76
Quick Guides	76
Quick Guide: How to save a view	76
Quick Guide: How to retrieve a view	76



Quick Guide: How to delete a view	76
Model Selection	77
Introduction	77
Interface Description	77
Quick Guides	77
Quick Guide: Using multiple model selection	77
Quick Guide: Solving overlapping models	77
Part Selection	78
Introduction	78
Interface Description	78
Quick Guides	78
Quick Guide: How to use the Part Selection Interface	78
Page 2:Pre/Post Processing	79
Introduction	79
CFD Time History Plot	80
Introduction	80
Interface Description	80
Quick Guides	80
Quick Guide: How to load a CFD time history component	80
Quick Guide: How to view a CFD time history plot	80
BINOUT	81
Introduction	81
Interface Description	81
Quick Guides	81
Quick Guide: How to Load and display data stored in a BINOUT file	81
CGAT – Circular Grid Analysis Technique	81
Introduction	82
Interface Description	82
Quick Guides	83
Quick Guide: How to form a grid of circles	83
Quick Guide: How to view time history data for a circle	83
Skid	84
Introduction	84
Interface Description	84
Quick Guides	85
Quick Guide: How to display a skid	85
Quick Guide: How to view time history data for a skid trace	85
General Selection Menu	86
Quick Guides	86
Quick Guide: Selecting nodes/elements/parts	86
Move or Copy Elements	87
Introduction	87
Interface Description	87
Quick Guides	87
Quick Guide: How to change an element's association with a part	87
Quick Guide: How to copy an element and apply an association with a part	87
Transform Model	88
Introduction	88



Interface Description	88
Quick Guides	88
Quick Guide: How to transform a model	88
Quick Guide: How to copy and transform a model	88
Translate Model	89
Introduction	89
Interface Description	89
Quick Guides	89
Quick Guide: How to Translate a model	89
Quick Guide: How to Translate a model	89
Reflect Model	90
Introduction	90
Interface Description	90
Quick Guides	90
Quick Guide: How to reflect a model	90
Quick Guide: How to copy and reflect a model	90
Rotate Model	91
Introduction	91
Interface Description	91
Quick Guides	91
Quick Guide: How to rotate a model	91
Quick Guide: How to copy and rotate a model	91
Project Model	92
Introduction	92
Interface Description	92
Quick Guides	92
Quick Guide: How to project a model	92
Quick Guide: How to copy and project a model	92
Scale Model	93
Introduction	93
Interface Description	93
Quick Guides	93
Quick Guide: How to scale a model	93
Quick Guide: How to copy and scale a model	93
Shell/Segment/Tshell Normals	94
Introduction	94
Interface Description	94
Quick Guides	94
Quick Guide: How to display Shell/Segment/Tshell normals	94
Page 3: Keyword File Editing	95
Introduction	95
Keyword Selection Menu	96
General Keyword Options	96
*CONTROL Extra Keyword Options	96
*EOS / *HOURLASS / *MAT Extra Keyword Options	96
*MAT Extra Keyword Options	96
Ref By Interface	97
Keyword Input Dialog	98
Quick Guides	98



Quick Guide: How to create a keyword using Page 3	98
SET Data	99
Quick Guides	99
Quick Guide: Entering Set Data	99
Link Keyword Dialog	100
Pick interface	100
Quick Guides	100
Quick Guide: How to pick data from the model	100
Dynamic Keyword Input Forms	101
Define Table	102
Quick Guides	102
Quick Guide: How to use define_table	102
Entity Draw Option	103
Introduction	103
Interface Description	103
Quick Guides	103
Quick Guide: How to display an entity from a keyword input form	103
Page 4: Pre Processing Tools	104
Introduction	104
Airbag Folding	105
Introduction	105
Interface Description	105
Quick Guides	106
Quick Guide: Creating an airbag fold	106
Quick Guide: How to simulate all folds	107
Quick Guide: How to simulate a single fold	107
Quick Guide: Animating folds	107
Dummy Positioning Interface	108
Introduction	108
Interface Description	108
Rotation Interface	109
Translation Interface	109
Mirror Interface	110
Limb Operations interface	110
Limb Orientation interface	111
Joint setting interface	111
File Write interface	112
Quick Guides	112
Quick Guide: How to load an occupant for dummy positioning	112
Quick Guide: How to rotate the occupant about the H-point	112
Quick Guide: How to translate the occupant	113
Quick Guide: How to mirror the occupant	113
Quick Guide: How to rotate limbs	113
Entity Operation Interface	114
Introduction	114
Interface Description	114
Using Show	114
Quick Guides	114



Quick Guide: How to Show entities on the drawing area	114
Quick Guide: How to Blank entities on the drawing area	115
Quick Guide: How to Identify entities on the drawing area	115
Quick Guide: How to use KCard	115
Using Modify/Delete	116
Quick Guides	116
Quick Guide: How to Select entities	116
Quick Guide: How to use KCard	116
Set Data	117
Introduction	117
Interface Description	117
Quick Guides	118
Quick Guide: Different ways of displaying set data	118
Quick Guide: How to create a Set	118
Quick Guide: How to modify existing Set data	118
Quick Guide: How to delete existing Set data	118
Quick Guide: How to write out Set data	119
*Define_Box Data	120
Introduction	120
Interface Description	120
Quick Guides	121
Quick Guide: Different ways of displaying Box data	121
Quick Guide: How to create a Box	121
Quick Guide: How to modify existing Box data	122
Quick Guide: How to delete existing Box data	122
*Define_Coordinate Data	123
Introduction	123
Interface Description	123
Interface Description	123
Coordinate Type Options	123
Quick Guides	124
Quick Guide: Different ways of displaying Coordinate data	124
Quick Guide: How to create Coordinate data	124
Quick Guide: How to modify existing Coordinate data	124
Quick Guide: How to delete existing Coordinate data	125
*Constrained_Generalized_Weld_Spot Data (GeWeld)	126
Introduction	126
Interface Description	126
Quick Guides	127
Quick Guide: Different ways of displaying Constrained Generalized Weld Spot data	127
Quick Guide: How to create a Constrained Generalized Weld Spot	127
Quick Guide: How to modify existing Constrained Generalized Weld Spot data	127
Quick Guide: How to delete existing Constrained Generalized Weld Spot data	127
*Constrained_Spot_Weld Data (SpWeld)	128
Introduction	128
Interface Description	128
Quick Guides	128
Quick Guide: Different ways of displaying Constrained Spot Weld data	128
Quick Guide: How to create a Constrained Spot Weld	129
Quick Guide: How to modify existing Constrained Spot Weld data	129
*Constrained_Spot_Weld Data (SpWeld)	130



Introduction	130
Interface Description	130
Quick Guides	130
Quick Guide: Different ways of displaying Constrained Spot Weld data	130
Quick Guide: How to create a Constrained Spot Weld	131
Quick Guide: How to modify existing Constrained Spot Weld data	131
Quick Guide: How to delete existing Constrained Spot Weld data	132
*Constrained_Spot_Weld Data (SpWeld)	133
Introduction	133
Interface Description	133
Quick Guides	133
Quick Guide: Different ways of displaying Constrained Spot Weld data	133
Quick Guide: How to create a Constrained Spot Weld	134
Quick Guide: How to modify existing Constrained Spot Weld data	134
*Constrained_Nodal_Rigid_Body Data (CNRB)	135
Introduction	135
Interface Description	135
Quick Guides	135
Quick Guide: Different ways of displaying CNRB data	135
Quick Guide: How to create a CNRB	135
Quick Guide: How to modify existing CNRB data	136
Quick Guide: How to delete existing CNRB data	136
*Initial_Velocity Data (IniVel)	137
Introduction	137
Interface Description	137
Quick Guides	137
Quick Guide: Different ways of displaying Initial Velocity data	137
Quick Guide: How to set an Initial Velocity	137
Quick Guide: How to modify existing Initial Velocity data	138
Quick Guide: How to delete existing Initial Velocity data	138
SPC Data	139
Introduction	139
Interface Description	139
Quick Guides	139
Quick Guide: Different ways of displaying SPC data	139
Quick Guide: How to create an SPC keyword	139
Quick Guide: How to modify SPC data	140
Quick Guide: How to delete existing SPC data	140
Constrained Rivet Data (Rivet)	141
Introduction	141
Interface Description	141
Quick Guides	141
Quick Guide: Different ways of displaying Constrained Rivet data	141
Quick Guide: How to create a Constrained Rivet	142
Quick Guide: How to modify existing Constrained Rivet data	142
Quick Guide: How to delete existing Constrained Rivet data	142
Constrained Node Set Data (ConstN)	143
Introduction	143
Interface Description	143
Quick Guides	143
Quick Guide: Different ways of displaying Constrained Node Set data	143



Quick Guide: How to create a Constrained Node Set	144
Quick Guide: How to modify existing Constrained Node Set data	144
Quick Guide: How to delete existing Constrained Node Set data	144
Database History Data	145
Introduction	145
Interface Description	145
Quick Guides	145
Quick Guide: Different ways of displaying Database History data	145
Quick Guide: How to create Database History data	146
Quick Guide: How to modify existing Database History data	146
Quick Guide: How to delete existing Database History data	146
Mass Data	147
Introduction	147
Interface Description	147
Quick Guides	147
Quick Guide: How to create Mass nodes	147
Quick Guide: How to modify Mass nodes	147
Part Data	148
Introduction	148
Interface Description	148
Quick Guides	150
Quick Guide: How to create Parts	150
Quick Guide: How to modify Parts	150
Quick Guide: How to use Search	150
Quick Guide: How to assign new keyword values to multiple parts	150
Page D: Display Interface	151
Introduction	151
Interface Description	151
Boundary	151
Constrained	151
Contact	152
Define	152
Initial	152
Load	152
Set	153
Seat Belt	153
Rigid Wall	153
Quick Guides	153
Quick Guide: Displaying entities using Page D	153
Appendix I	154
Configuration File Options	154
Introduction	154
List of Configuration File Options	154



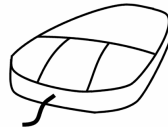
KEYBOARD AND MOUSE OPERATION

DYNAMIC MODEL OPERATION

Rotation – Left mouse button + shift key

Translation – Middle mouse button + shift key

Scaling – Right mouse button + shift key.



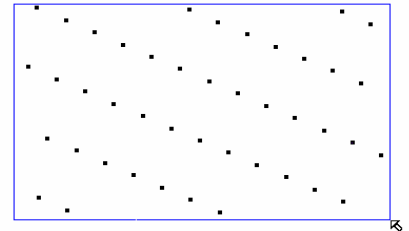
Fast rendering mode – Use Ctrl key instead of shift key, this will render the model in edge mode.

DRAWING AREA SELECTION

Single Pick – Left mouse button

Area Pick or windowing – Left mouse click, hold and drag.

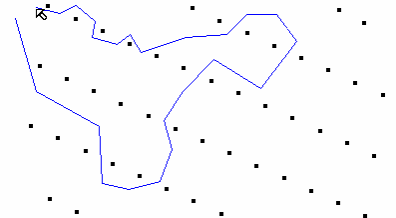
Polygon pick – Left mouse click, click point to form each side. Right click to complete.



LISTS AND MULTIPLE SELECTIONS

If multiple selections are allowed then the following will apply.

1. Press and hold Ctrl key
2. Make multiple selections





DROP DOWN MENUS

FILE

New – Close all currently loaded models and prepare to load a new one

Open – File chooser to select and open files

Update – while LSDYNA is outputting d3plot files u can update

Save Keyword – Write pre processor data to a keyword file (*.k)

Save Config – Write configuration file (.lsposrc)

Print – Activate print dialog (see below for further details)

Movie – Activate movie creation dialog (see below for further details)

Exit – Exit LS Pre/Post v1.0

Save_Exit – Save current data and Exit LS Pre/Post v1.0



OPEN

Binary Plot – Open a binary plot file for post analysis

Time History – Open a time history data file

Command File – Open a (.Cfile)

Database File – Open a (.db) file

Message File – Open a (.msg) file

Crack – Open a crack file

Config File – Open a configuration file

FLD Curve – Open a FLD curve data file

Labels – Open a labels file

Xydata – Open a XY-Plot data file

Background – Load a background image

Keyword – Load a (.k) file

Interface Force – Open LS-DYNA interface force file

Nastran File – Open a Nastran file





PRINT DIALOG

Name – Select printer

Properties – Select printer settings

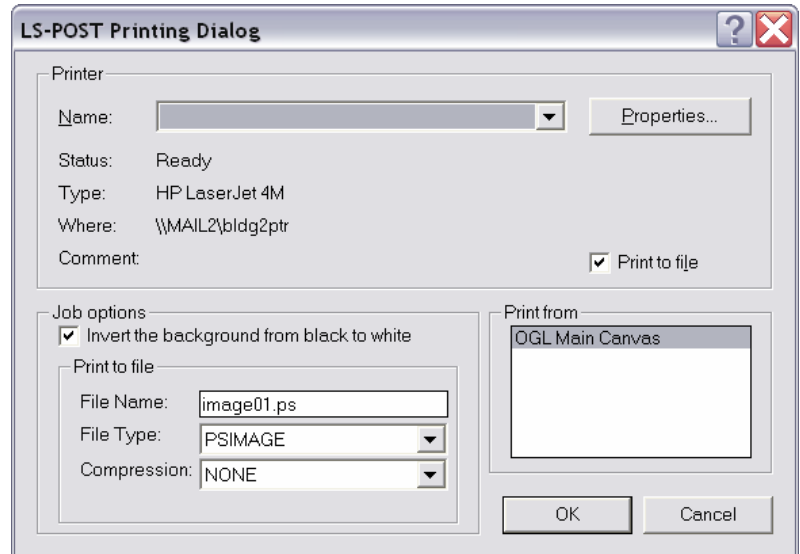
Print to file – Select if data is to be outputted to a file

File Name – Input file name

File Type – Select file format

Compression – Select file compression format

Print from – Select data to print



Quick Guides

Quick Guide: Printing drawing area to a file

1. Set model in position in drawing area
2. Open Printing Dialog
3. Select Print to file
4. Set background color
5. Enter file name
6. Select image format
7. Select compression format
8. Click [OK]

Quick Guide: Printing drawing area to paper

1. Set model in position in Drawing area
2. Open Printing Dialog
3. Select printer (note a printer must already be setup with the pc)
4. Modify printing properties
5. Deselect Print to File
6. Click [OK]



DROP DOWN MENUS

MOVIE DIALOG

Format – Select movie file format

Repeat – Set number times to repeat animation in movie

Use Playback File – Use command_record data file during movie generation

Size – Select movie resolution

Window size – Enter movie resolution

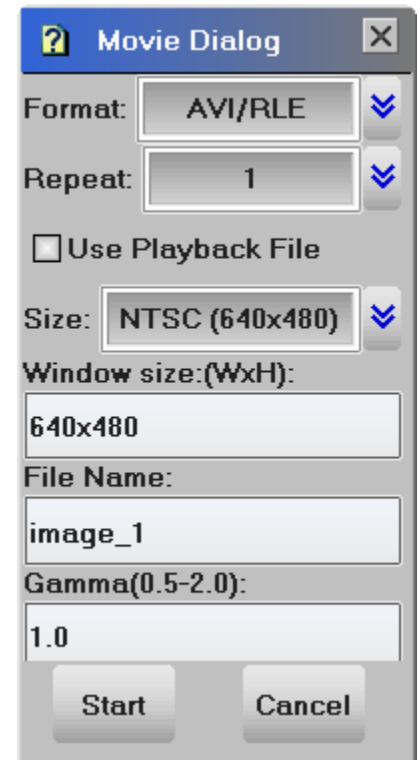
File Name – Enter movie filename

Gamma – Enter gamma correction

Quick Guides

Quick Guide: How to create a movie

1. Open a model
2. Select movie file format
3. Select number of times to record animation
4. Check Use Playback to generate movie of a pre recorded set of commands. (See Command File for details, note the command history must be recorded using the options under the Misc. menu)
5. Select movie format and size or enter a custom size
6. Enter file name
7. Enter gamma correction
8. Click [Start]



MISC.

Reflect – Reflect model about a global plane

Model Info – Give model size information

Swap byte on Title – Show title in a different byte order

Mesh line width – Specify mesh line width in pixels

Edge line width – Specify edge line width in pixels

Feature angle – Select feature line angle (if the angle between 2 elements is greater than the feature line angle a line will drawn).

Start record – Begin recording command file data

Stop record – Stop recording command file data

Playback – Playback recently recorded command file data (command_record in working directory)

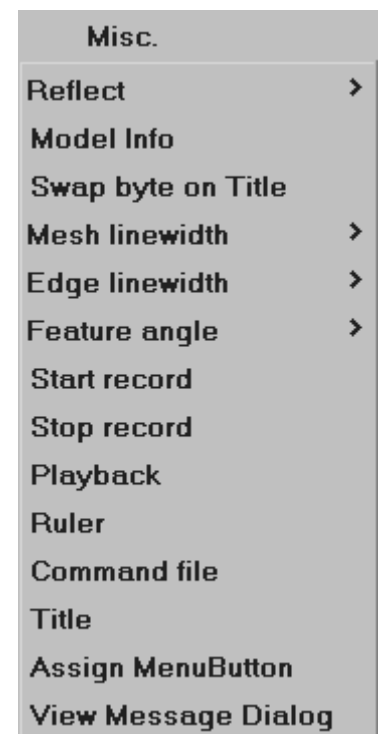
Ruler – Opens ruler dialog (see below for further details)

Command file – Open command file dialog

Title – Modify model title

Assign Menu Button – Open assign menu button dialog (see below for further details)

View Message Dialog – Open keyword file read error message dialog





DROP DOWN MENUS

TOGGLE

In the following the first option is the default option.

The setting displayed in the menu indicates the next setting not the current setting.

Local axes – off/on

Texture –switch texture mode off/on (only applicable when in Fringe mode)

Lighting – Two-sided/One-sided

Fringe Light – off/on When this option is off the model will be in fringe color mode. (Lighting effects are disabled)

Outlining – Edge/feature/off, select outline type.

PlotUnode – Toggle plot un-referenced nodes on/off

Highlight – Toggle highlights off/on

Result On Screen – Toggle Show results on screen on/off (only applicable if show results is selected under Identify interface)

Shade – Flat/smooth

Outline – Select outline color black/white

Deleted Nodes – on/off

Record Message – off/on When on all messages displayed in the command window will be saved to file, Default file = lspost.msg

Beam Style – Line/Prism

Animate mode – Loop/Once

Deleted elements – off/on



BACKGROUND

Full – Show drawing area in full screen mode

Plain – Set single color background (Select color from color interface)

Fade – Set two color faded background (Select color from color interface)

DiaFade – Set two color double faded background (Select color from color interface)

TriFade – Set three color faded background (Select color from color interface)

Picture – Set jpeg as background (Use File/Open to import jpeg image first)



HELP

Help – Not implemented

Label – Toggle floating help bubbles off/on.





COMMAND FILE (CFILE) DIALOG

Introduction

Accessible via the Misc. or File/Open drop down menu. A Cfile is a log of every command executed during a program session. This log can be viewed using a normal text editor, or replayed within LS Pre Post. CFiles are saved in the same directory as the program executable.

Dialog Description

Pause – Time delay between displaying commands

Loop continuously – Loop Cfile playback

Open – Use to open a Cfile if the original one is closed

Start – Start playback

Stop – Stop playback

Step – Step through a single command at a time

Rewind – Return to the beginning of the command file

Close – Close current Cfile

Done – Exit Cfile Dialog

Quick Guides

Quick Guide: How to record a command file

1. Click Misc./Start recording
2. Proceed to carry out commands to record
3. Click Misc./Stop recording
4. The recorded commands are saved as 'command_record' in the same directory as the current model.
Or Copy/Rename lspost.cfile found in the same directory as the program executable. Note this file contains command data for the complete session.

Quick Guide: How to replay a command file

1. Click Misc./Playback to replay a recently recorded command file
Or Click File/Open/Command File, Select the Cfile and click [Open]
2. Select pause time between each command
3. Click [Start] to begin replay





RULER

Introduction

Display ruler(s) in drawing area.

Dialog Description

Ruler – Select ruler type

Line Width – Select ruler line width

Color – Select ruler line color

Horizontal Ruler – Adjust horizontal ruler position

Vertical Ruler – Adjust vertical ruler position

[Reset] – Reset rulers to default settings

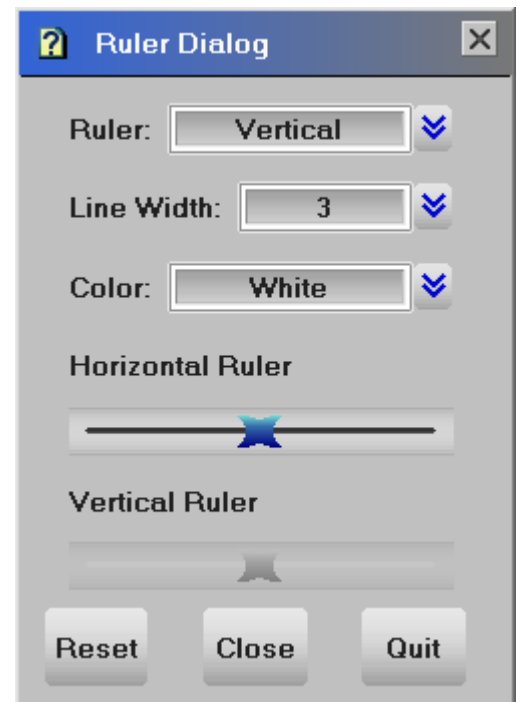
[Close] – Close ruler dialog

[Quit] – Close ruler dialog and remove ruler(s)

Quick Guides

Quick Guide: How to use the ruler

1. Select Misc./Ruler to open dialog
 2. Select rulers to display
 3. Select ruler line width
 4. Select ruler line color
 5. Position ruler using slider
 6. Click close to close dialog and keep the rulers
- Or
7. Click Quit to close the dialog and remove the rulers
- (Note: A model must be open to use rulers)





ASSIGN MENU BUTTONS

Introduction

This dialog allows the manipulation of interface buttons on the right hand menu.

Dialog Description

List of available interfaces

[Save] – Save button assignments to .lsposrc file

[Reset] – Reset main menu to default button assignments

[Clear] – Clear all assignments on selected page

[Done] – Close dialog (Note any changes made without saving will remain for the current session only)

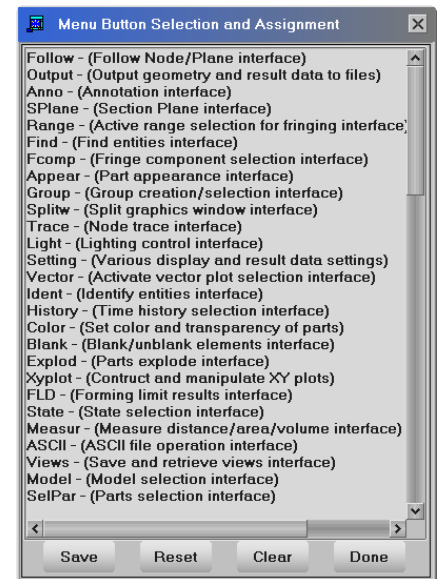
Quick Guides

Quick Guide: How to Assign menu buttons

1. Open assign menu button dialog
2. Select button for assignment (does not have to be empty)
3. Select interface from the list to assign to the selected button
4. Click [Save] to save changes in .lsposrc file

Quick Guide: How to clear menu button assignments

1. Open assign menu button dialog
2. Select button for clearing
3. Click [Clear]





COMMAND INTERFACE

Finished reading model

1

open d3plot "D:\models\draw\d3plot"
Reading binary plot files
Finished reading model

2

Last Command Text Field– The last command issued is displayed here(1)

Previous command list – All issued commands are shown in this list. Also, if a command is issued with missing criteria an error message may be displayed here. (2)

(Left click in the Last Command Text Field to open, left click again to close.)

>

Command line input – Commands may be entered here when a help comment is not being displayed.

Auto center the model to the window

Help Comments – These are displayed in the command line input area when the mouse cursor is placed over an interface. Further help may be shown in the previous commands list.



ANIMATION CONTROL

First	1	Last	44	Inc	1		SF	1.0	Time	0	
							0	No.of Div	1	State#	1
									Perf:	0.01 s/f	

First – Enter initial state to display

Last – Enter final state to display

Inc – Enter number of states to increment per displayed frame

Time –Time of current state

State – Enter state number to display

Perf – Time per frame



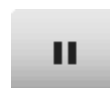
Animate Forwards and Backwards



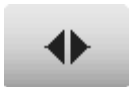
Stop Animation



Increment Animation Forwards and Backwards



Slider will change state only



Reverse animation direction



Slider will Animate and change State



Slider



Speed Up

0

Animation Speed Control

Slow Down

SF	1.0
No.of Div	1

Enter Displacement Scale Factor

Number of intermediate steps between states
(For EIGEN value analysis this is the number of steps in each mode)



RENDERING HOT BUTTONS

Title	Legd	Tims	Triad	Bcolr	Mcolr	Frin	Isos	Lcon	Acen	Zin	+10	Rx	DeOff	DeOn	Top	Front	Right	Redw	Home
Hide	Shad	View	Wire	Feat	Edge	Grid	Mesh	Shrn	Pcen	Zout	//	Clp	All	Rpat	Bottom	Back	Left	Anim	Rset

Title	Legd	Tims	Triad	Bcolr	Mcolr
-------	------	------	-------	-------	-------

Title – Toggle showing title on/off

Legd – Toggle showing legend on/off

Tims – Toggle showing time stamp on/off

Triad – Toggle showing triad on/off

Bcolr – Toggle background color black/white

Mcolr – Toggle mesh line color black/white

Frin	Isos	Lcon
------	------	------

Frin – Color fringe contour

Isos – Iso-surface plot, (applicable to 3D solid elements only)

Lcon – Color line contour

Hide	Shad	View	Wire	Feat	Edge	Grid	Mesh	Shrn
------	------	------	------	------	------	------	------	------

Hide – Display model with hidden line removed

Shad – Display model in color shading mode
(Toggle flat/smooth using toggle menu)

View – Display model in plain color mode

Wire – Display model as a wire frame

Feat – Display model in Feature line mode (default
angle =30 deg)

Edge – Display model in edge line mode

Grid – Display each nodal point as a colored pixel

Mesh – Overlay mesh line on shade or color plot

Shrn – Draw elements in shrunk mode (default =
0.85)

Acen	Zin
Pcen	Zout

Acen – Automatically center model to fit within window

Pcen – Pick a node to be the new center point for model rotation

Zin – Zoom in, draw a box using the left mouse button and dragging

Zout – Zoom out to previous zoom position

+10	Rx
-----	----

+/- 10, Rx, Ry, Rz – Rotate 10 degrees about the global X, Y, Z axes respectively. Right click on Rx, Ry, Rz to switch rotation axis.

//

Parallel view enabled,
click to switch to perspective view

Pers

Perspective view enabled,
click to switch to parallel view



RENDERING HOT BUTTONS

DeOff

DeOn

DeOff – Turn off all displayed entities

DeOn – Turn on all previously displayed entities

Clp

All

Rpat

Clp – Clear all picked or highlighted information

All – Restore all entities to be active

Rpat – Restore the last removed part

Top

Front

Right

Bottm

Back

Left

Switch to labeled view

Redw

Home

Anim

Rset

Redw – Re-draw current model

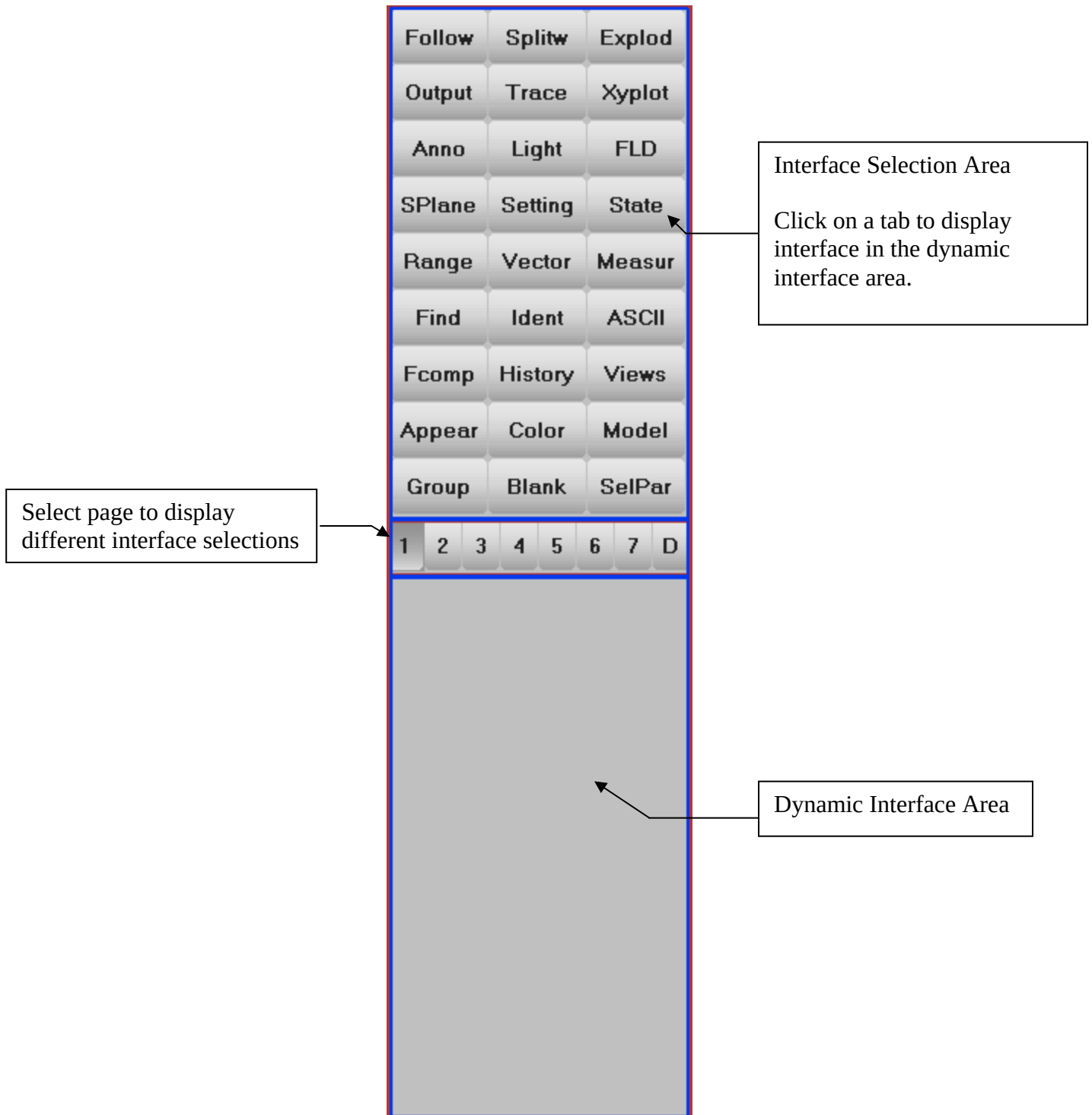
Home – set model in home position

Anim – Start or Stop Animation

Rset – Restore model to original position and state



MAIN MENU





PAGE 1:POST PROCESSING

INTRODUCTION

The interfaces found on this page are for general model manipulation and post processing.

Follow	Splitw	Explod
Output	Trace	Xyplot
Anno	Light	FLD
SPlane	Setting	State
Range	Vector	Measur
Find	Ident	ASCII
Fcomp	History	Views
Appear	Color	Model
Group	Blank	SelfPar



FOLLOW OPTIONS

Introduction

This interface is used in conjunction with model animation. By defining a single point or a plane, this will be displayed in the center of the screen at all times during the model animation.

Interface Description

Follow Point – Pick node to follow

Follow Plane – Pick n1, n2 and n3 to follow a plane

Node 1, 2 and 3 – Enter node number for nodes 1, 2 and 3 respectively

All – Follow plane with regards to whole model

Part – Follow plane with regards to selected parts. Pick/Enter part ID

[Apply] – Apply follow to model view

[Clear] – Clear pick list

[Reset] – Deactivate follow mode and restore model to previous position

[Done] – Exit Follow Options interface

Quick Guides

Quick Guide: How to follow a node

1. Click [Follow] to open interface
2. Select *Follow Point*
3. Pick/Enter 1 node ID
4. Click [Apply]

Quick Guide: How to follow a plane

1. Click [Follow] to open interface
2. Select *Follow Plane*
3. Pick/Enter 3 node IDs
4. Click [Apply]

Follow Options

☒ Follow Point

☐ Follow Plane

Define Plane:

☒ Node1

☐ Node2

☐ Node3

☐ Part

☒ All

Apply Clear

Reset Done



FORMATTED OUTPUT

Introduction

Use this interface to output model data.

Interface Description

Format – Select output style

Active parts only – Write out active parts and elements only

Entire Model – Write data for the entire model

Internal Number – Write elements and nodes using internal numbers

User Number – Write elements and nodes using user numbers

Element – Write element connectivity to file

Node coordinate – Write nodal coordinates to file

Node displacement – Write nodal displacement to file

Element results – Write element results to file

Nodal results – Write nodal results to file

Shell P-strain – Write shell principal strain to file

State No: – Enter state sequence to be written (e.g. 1:5:2 would write out data for states 1, 5 and 2 in that sequence)

Current – Set State No: to current state

Append – Select to append data to an existing file

Write – Start writing file

Done – Exit Formatted output interface

Quick Guides

Quick Guide: How to write an output data file

1. Select output format
2. Select elements and parts for output
3. Select the ID format
4. Select additional data output options
5. Enter state sequence for output
6. Select *Append* to append an existing file
7. Click [Write] to begin data output

The screenshot shows the 'Formatted Output' dialog box. At the top, there is a 'Format:' dropdown menu currently set to 'Keyword'. Below this are two radio buttons: 'Active parts only' (which is selected) and 'Entire Model'. A horizontal line separates these from the next section, which contains two radio buttons: 'Internal ID' and 'User ID' (which is selected). Another horizontal line follows. Below that is a list of checkboxes: 'Element', 'Nodal Coordinates', 'Nodal Displacement', 'Element Results', 'Nodal Results', and 'Shell P-Strain'. At the bottom left, there is a 'State no:' label next to a text input field. Below the input field is an 'Append' checkbox. To the right of the 'Append' checkbox are two buttons: 'Current' and 'Write'. At the bottom right is a 'Done' button.



ANNOTATION

Introduction

Use this interface to add annotations to the model.

Interface Description

Select label from list (1)

[Add] – Add new label to list

[Clr] – Clear label input field

[Del] – Remove selected label from list

Text – Enter new label to add to list (2)

Position – Position label in drawing area

Move – Move label in drawing area

Select – Select label in drawing area

Arrow – Position label arrow in drawing area

[Del] – Delete selected labels in drawing area

[Dall] – Delete all labels in drawing area

[Set] – Function no longer required

[Font] – Function no longer required

Col – Select label and arrow color

Ang – Select label angle

Pts – Select label font point size

Font – Select basic font for program

[Reset] – Delete and clear all labels

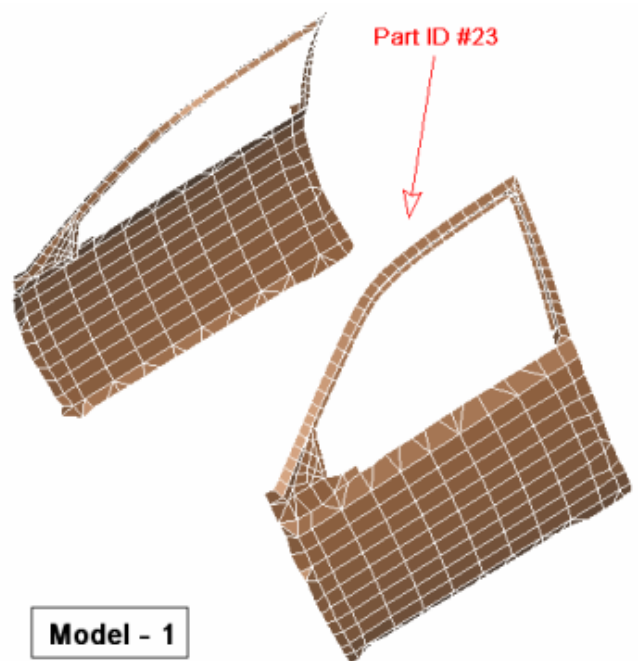
[Done] – Exit Annotation interface

The screenshot shows the 'Annotation' interface. At the top, it displays 'Model - 1' and 'Part ID #23', with a callout '1' pointing to this text. Below this is a list of labels, currently empty, with a callout '2' pointing to it. To the right of the list are buttons for 'Add', 'Clr', and 'Del'. Below the list is a 'Text:' input field. Further down are radio buttons for 'Position' (selected), 'Move', 'Select', and 'Arrow'. Below these are buttons for 'Del', 'Dall', 'Set', and 'Font'. At the bottom are input fields for 'Col' (a color square), 'Ang' (a dropdown showing '0'), 'Pts' (a dropdown showing '14'), and 'Font' (a dropdown showing 'Helvetica'). At the very bottom are 'Reset' and 'Done' buttons.



Quick Guide: How to add a label with an arrow

1. Enter new label
2. Click [Add]
3. Select label from list
4. Select label and arrow color
5. Select label angle
6. Select label font size
7. Select label font
8. Select *Position* from interface
9. Left click in drawing area to position label
10. Select *Select* from interface
11. Left click on positioned label
12. Select *Arrow* from interface
13. Left click on an existing label, hold and drag to create the arrow





SECTION PLANE

Introduction

Use this interface to create section planes that can be used to perform section cuts of the model for further analysis.

Interface Description

FixSp – Section Plane is fixed in space

FixMd – Section Plane is fixed to the model

1p+NL – Define plane with 1 point and a normal

BasePt – Define base point by entering x, y, z coordinates

BaseNd – Define base point by picking/entering a node ID

X, Y, Z – Enter respective base point coordinates

Node – Enter node number for base point

Norm X, Y, Z – Enter respective components of normal vectors

[Centroid] – Use centroid of model as base point

[Reset] – Clear base point and normal settings

Quick Guides

Quick Guide: How to define a plane using 1p+NL

1. Enter coordinates
Or select *BaseNd* and pick a node from the model
Or click [Centroid] to use center of model as base point
2. Enter normal vector or use X/Y/Z axes by clicking on the buttons [NormX/Y/Z]

The screenshot shows the 'Section Plane' dialog box. It has a title bar with a right-pointing arrow and the text 'Section Plane'. Below the title bar are two radio buttons: 'FixSp' (selected) and 'FixMd'. Below these are two more radio buttons: '1p+NL' (selected) and 'N1-N2'. Below those are two more radio buttons: '3Nds' and '2Nds+D'. Below these are two checkboxes: 'BasePt' (checked) and 'BaseNd' (unchecked). Below the checkboxes are three input fields for 'X:', 'Y:', and 'Z:', with values '10', '15', and '20' respectively. Below these is a 'Node' input field. Below the 'Node' field are three buttons: 'NormX', 'NormY', and 'NormZ'. Below these are three input fields for '0.0', '0.0', and '1.0' respectively. Below these are two buttons: 'Centroid' and 'Reset'.



N1-N2 – Define point normal by picking 2 nodes
3Nds – Define point normal by picking 3 nodes
Node 1, 2 and 3 – Enter or pick a respective node number
2Nds+D – Define plane by picking 2 nodes and a direction cosine
Dcos X, Y and Z – Enter respective direction cosine components

[Reset] – Clear all picked nodes

Quick Guides

Quick Guide: How to define a plane using N1-N2

1. Pick 2 nodes from the model

Note: Node 1 is a point on the plane and N1-N2 is a direction vector

Quick Guide: How to define a plane using 3Nds

1. Pick 3 nodes from the model

Note: All 3 nodes will be on the plane

Quick Guide: How to define a plane using 2Nds+D

1. Pick 2 nodes from the model
2. Enter a direction cosine

→Section Plane←

☒ FixSp ☐ FixMd

☐ 1p+NL

☐ N1-N2

☐ 3Nds

☒ 2Nds+D

Define Plane:

☒ Node1

☐ Node2

☐ Node3

DcosX

DcosY

DcosZ

0.0

0.0

1.0

NormX

NormY

NormZ

0.00

0.00

0.00

Reset



[Cut] – Cut the section

[Model] – Display normal model

[Force] – Open section force plot interface

[Mvp] – Move section plane along normal direction

[Options] – Other options or section plane

[Crush] – Plot intrusions relative to a plane

[Line] – Open section cut line plot interface

[Done] – Exit Section Plane interface



Force: Section force plot interface

Select items for plotting (1)

[Plot] – Plot section variable vs time in current XY-Plot window

[New] – Plot section variable vs time in a new XY-Plot window

[Padd] – Add selected data to current XY-Plot window

[Raise] – Raise all open XY-Plot windows

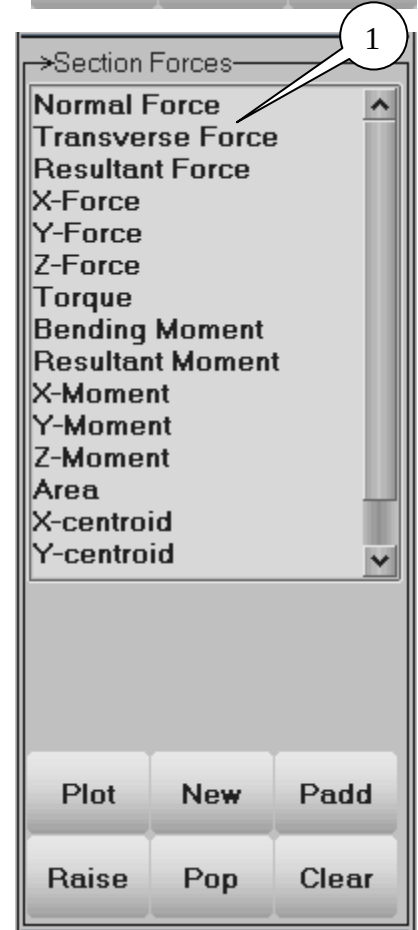
[Pop] – Open and Raise all closed XY-Plot windows

[Clear] – Clear selected items in the list

Quick Guides

Quick Guide: How to display a force plot

1. Define a plane
2. Click [Force]
3. Select a section force
4. Click [Plot]/[New]





MVP: Moving Section Plane Interface

of Cuts – Enter number of cuts to make in model

Distance – Enter distance to move plane along the normal



Step plane backwards or forwards in direction of the normal



Use slider to move plane in direction of the normal

Keep all Sections – Store all section cuts in memory

Keep all Cuts – Display all section cuts in drawing area

Mp anim – Animate moving plane along the normal

Update Basept – Update base point with current position

Quick Guides

Quick Guide: How to setup a moving plane

1. Click [MVP]
2. Select [# of Cuts] and enter number of cuts to make in model
3. Select [Distance] and enter distance to move plane from origin
4. Use slider or arrows to move section plane or click [mp anim] to animate

Quick Guide: How to store and view multiple section cuts

1. Setup a moving plane as described above
2. Check Keep all sections (Each section cut made from now on will be stored)
3. Once all cuts have been stored check Display all sections (All currently stored sections will be displayed)

Base Pt. Location:

Xpos: 50

Ypos: 25

Zpos: 2.5

#of Cuts 50

☐ Keep all Sections

mp anim

Update Basept



Options: Section Plane Options

ShowPl – Show section plane with model

ShMesh – Show section plane as mesh

Enter grid size for mesh (1)

Outline – Show 3D model outline with section

LnWidth – Set section cut line width

LnColor – Set section cut line color

File Writing

State No. – Enter state number for the section data to be written

Curr State – Write section data for current state

Write – Start writing section data to file

Quick Guides

Quick Guide: How to output section cut data

1. Define a section plane
2. Enter state no. / click [Curr State]
3. Click [Write]
4. Enter filename
5. Click [Save]

☒ ShowPl ☐ Outline

☐ ShMesh 7

LnWidth 1

LnColor Partcolor

state no:

Write Curr State



Crush: Intrusion Plotting

LnWidth – Select intrusion plotting line width

LnColor – Select intrusion plotting line color

Plot Type – Select history plot type

Node ID – Enter or pick a node ID to compute intrusion
(Hit Enter key to accept an entered node ID)

[Plot] – Plot intrusion line data in current XY-Plot window

[New] – Plot intrusion line data in a new XY-Plot window

[Padd] – Add intrusion line data to current XY-Plot window

[Raise] – Raise all open XY-Plot windows

[Pop] – Open and Raise all closed XY-Plot windows

[Clear] – Clear picked/entered node IDs

Quick Guides

Quick Guide: How to display an intrusion plot

1. Click [Crush]
2. Select line width/color
3. Select plot type
4. Pick/Enter node IDs
5. Click [Plot]/[New]

Intrusion Plotting:

LnWidth: 1

Line Color: White

PlotType: Crush

Node ID:

Plot New Padd

Clear Raise Pop



Line: Cut Line Plot interface

[Plot] – Plot section distance vs fringe component in current XY-Plot window

[New] – Plot section distance vs fringe component in a new XY-Plot window

[Padd] – Add section distance vs fringe component data to current XY-Plot window

[Raise] – Raise all open XY-Plot windows

[Pop] – Open and Raise all closed XY-Plot windows

[Flip] – Flip the XY curve in the x-axis

Quick Guides

Quick Guide: How to plot Cut Line data

1. Select a Fringe Component using Fcomp (see Fringe Component for details)
2. Define a section plane and perform a cut
3. Click [Line] to open Cut Line interface
4. Pick a part
5. Click [Plot]/[New]

Selected PartID:	45	
Coord at X = 0:		
X=	6301.936035	
Y=	-219.145782	
Z=	848.104980	
Coord at X = End:		
X=	7139.279785	
Y=	-1382.768921	
Z=	848.104980	
Plot	New	Padd
Flip	Raise	Pop



SET FRINGE RANGE

Introduction

Use this interface to set fringe and Iso-surface ranges.

Interface Description

Dynamic – A set of min/max ranges is computed for each time state

Static – A constant min/max range is computed using all time states

User – Range set by user, enter min/max values below

Show – Show elements within the range entered below

Entire Model – Range computed for entire model

Active Part only – Range computed for active parts only

Active Ele only – Range computed for active elements only

Blank out of range – Blank all elements with values outside the user defined range

No average – Fringe value is not averaged at node

Show active min/max – Show min and max values for current view

Set Isos Range – Set Iso-surface Range values independently

Reverse Palette – Reverse the color palette

Ident Min value – Identify first N minimum values

Ident Max value – Identify first N maximum values

No. of min/max entities – Enter number of min/max values to be identified

Levels – Select or enter number of color in the fringe palette

[Palette] – Open fringe color palette

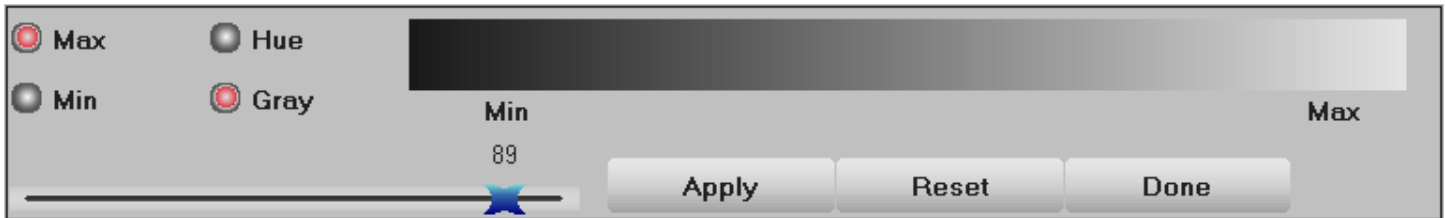
[Update] – Update range settings

[Done] – Exit Set Fringe Range interface

The screenshot shows the 'Set Fringe Range' dialog box. It has a title bar with a right-pointing arrow and the text 'Set Fringe Range'. The interface is organized into several sections. The top section contains radio buttons for 'Dynamic' (selected), 'Static', 'User', and 'Show'. Below this is a section with radio buttons for 'Entire Model', 'Active parts Only' (selected), and 'Active Elements Only'. The next section has two input fields labeled 'Min:' and 'Max:'. Below these are several checkboxes: 'Blank out of range', 'No Average', 'Show active min/max', 'Set Iso Range', 'Reverse Palette', 'Ident Min value', and 'Ident Max value'. At the bottom, there is a label 'No. min/max entities' with a text box containing the number '5'. Below that is a 'Levels' section with a text box containing '10', a blue double-checkmark icon, and another text box containing '10'. At the very bottom are three buttons: 'Palette', 'Update', and 'Done'.



Fringe Color Palette



Max – Set fringe color for maximum of range

Min – Set fringe color for minimum of range

Hue – Use slider bar to set Hue color value

Gray – Use slider bar to set Gray shade value

[Apply] – Apply current palette to the fringe plot

[Reset] – Reset fringe palette to default values

[Done] – Exit the Fringe Color Palette

Quick Guides

Quick Guide: How to set a fringe range

1. Select range type
(Min/Max values need to be entered for User and Show)
2. Select entities to be used for fringe viewing
3. Enable/Disable extra options
4. Enter number min/max entities to show in top left
5. Select/Enter number of fringe levels to be used.

Quick Guide: How to set the fringe color palette

1. Click [Palette] to open color palette
2. Select Max and use slider to set value
3. Select Min and use slider to set value
4. Select between hue / gray for palette color
5. Click [Apply] to store changes in database (db) file

Quick Guide: How to use Show

1. Select show
2. Enter min/max values (Hit enter after each entry to accept it)

Quick Guide: How to show active minima and maxima

1. Apply a fringe component to the model
2. Open Fringe Range interface
3. Check Show active min/max
4. Click [Zin]
5. Define area in drawing area
(Note: to be able to zoom in Show active min/max must be unchecked)



Find Ele/Node/Mat

Introduction

Use this interface to find a node/element/part by entering its unique ID.

Interface Description

Node – Find a node by entering a node ID

Elem – Find an element by entering an element ID

Part – Find a part by entering a part ID

Any – Find elements of any element type

Beam – Find beam elements

Shell – Find shell elements

Solid – Find solid elements

Tshell – Find Tshell elements

SPH node – Find SPH elements

Show only – Show the entity only

Highlight – Highlight the entity

Enter ID (1)

[Find] – Find and display the entity data

[Neighbors] – Find neighboring nodes/elements

[Done] – Exit Find interface

[Un blank All] – Un blank all blanked elements

[Un blank Part] – Un blank part of found elements

Quick Guides

Quick Guide: How to find an entity

1. Select entity type
2. Select entity sub class
3. Select how to display entity
4. Enter entity ID
5. Click [Find]

The screenshot shows the 'Find Ele/Node/Mat' interface. At the top, there are radio buttons for 'Node' (selected), 'Element', and 'Part'. To the right, there are radio buttons for 'Any' (selected), 'Beam', 'Shell', 'Solid', 'Tshell', and 'Sphnode'. Below these are two more radio buttons: 'Show Only' and 'Highlight' (selected). A text input field labeled 'Node/Elem/Part Id' is highlighted with a callout bubble containing the number '1'. At the bottom, there are four buttons: 'Find', 'Neighbors', 'Done', 'UnblankAll', and 'UnblankPart'.



FRINGE COMPONENT

Introduction

Use this interface to display fringe component data on the model.

Interface Description

- [Stress] – Global Stress/Strain components
- [Ndv] – Nodal Displacement/Velocity Contour
- [Result] – Stress resultant components
- [Strain] – Logarithmic strain components
- [Misc] – Pressure, Temperature, Thickness, etc.
- [Infin] – Infinitesimal Strains
- [Green] – Green-St. Venant Strains
- [Almans] – Almansi Strain
- [S.Rate] – Strain rates
- [Residue] – Residual elastic strains
- [FLD] – FLD strain components
- [Beam] – Beam fringe components
- [CFD] – Navier-Stokes components
- [D3Mean] – Time-average window fringe components
- [User] – User defined fringe components
- [Apply] –Collect fringe data



Model result type (MP)



- Fringe
- Iso-surface
- Line Contour
- Fringe Iso-surface
- XFringe – finding the max value across all surface



Set shell stress surface position



- Low
- Mid
- Upper
- Maximum
- Integration Point
- Beam Integration Point



Set axes for stress components



- Global Axes
- Local Axes





Quick Guides

Quick Guide: How to display fringe components

1. Select fringe component
2. Select model result type (note ISO Surface can only be used on solid elements)
3. Select shell stress surface position
4. Select axes Global/Local
5. Click [Apply]



SET PART APPEARANCE

Introduction

Use this interface to change the appearance of selected parts. This information can be saved using the Save/Retrieve Views interface.

Interface Description

Pick Part – Assign appearance by part

Area – Assign appearance to parts inside/outside a user defined area

Polygon – Assign appearance to parts inside/outside a user defined region

In – Set parts inside the area/polygon

Out – Set parts outside the area/polygon

Off – No change to the part display mode

Hide – Display part(s) with hidden line removed

Shad – Display part(s) in shaded mode

View – Display part(s) in non-shaded color mode

Wire – Display part(s) in wire frame mode

Feat – Display part(s) in feature line mode

Edge – Display part(s) in edge line mode

Grid – Display part(s) with grid points

Frin – Display part(s) with fringe

Mesh – Toggle mesh outlining on/off with shad and fringe

Shrn – Toggle shrink plot on/off

Isos – Display part(s) with iso-surfaces

Lcon – Display part(s) with line-contours

[AllVis] – Assign appearance to all visible parts

[Done] – Exit Set Appearance interface

Quick Guides

Quick Guide: How to set part appearance

1. Select appearance options
 2. Pick parts using pick/area/polygon
- Or click AllVis to apply to all visible parts

Set Part Appearance

☐ Pick Part

☒ Area ☒ In

☐ Polygon ☐ Out

☒ Off ☐ Feat

☐ Hide ☐ Edge

☐ Shad ☐ Grid

☐ View ☐ Frin

☐ Wire

☐ Mesh

☐ Shrn

☐ Isos

☐ Lcon

☐ Thick

AllVis Done



GROUP OPERATIONS

Introduction

Use this interface to create and manipulate groups of parts.

Interface Description

Group list – Select created group from list (1)

Group Name – Enter new group name (2)

[Create] – Create a new group from the active set

[Delete] – Delete selected group from the list

[Rename] – Re-name a selected group

[Select] – Make the selected group the active set

[Add] – Add the selected group to the active set

[Sub] – Subtract the selected group from the active set

[Save] – Save group data to a file

[Load] – Load group data from a file

[Done] – Exit Group Operations interface

Quick Guides

Quick Guide: How to create a group

1. Use Part Selection interface [SelPar] to select parts for group
2. Enter a name for the group
3. Click [Create]

Quick Guide: Available group operations

Or + [Add] – All parts from both groups are added together

Or + [Sub] – Parts in the highlighted non-active set are removed from the active set.

And + [Add] – Parts common to both groups are kept

And + [Sub] – Parts common to both groups are removed

Xor + [Add] – Parts not common to both groups are kept

Xor + [Sub] – Parts common to both groups are removed; only uncommon parts from the active group remain.

→Create/select Group

Group_01
Group_02

1

Group Name:

2

Group_03

Create	Delete	Renam
Select	Add	Sub
Save	Load	Done

☒ Or ☐ And ☐ Xor



SPLIT WINDOW

Introduction

Use this interface to open up to four drawing areas. Each window can be operated independently and can display different models. The windows can also be synchronized by state and all animated together.

Interface Description

- 1x1 – Single drawing area
- 1x2 – Split window horizontally (two drawing areas)
- 2x1 – Split window vertically (two drawing areas)
- 2x2 – Split window horizontally and vertically (four drawing areas)

Select model to display in active drawing area (1)

- Tleft – Draw to top left drawing area
- Bleft – Draw to bottom left drawing area
- Tright – Draw to top right drawing area
- Bright – Draw to bottom right drawing area

Draw all areas – Dynamic model operations will affect all drawing areas

[Sync] – Set all drawing areas (models) to state 1

[Done] – Exit Split Window interface

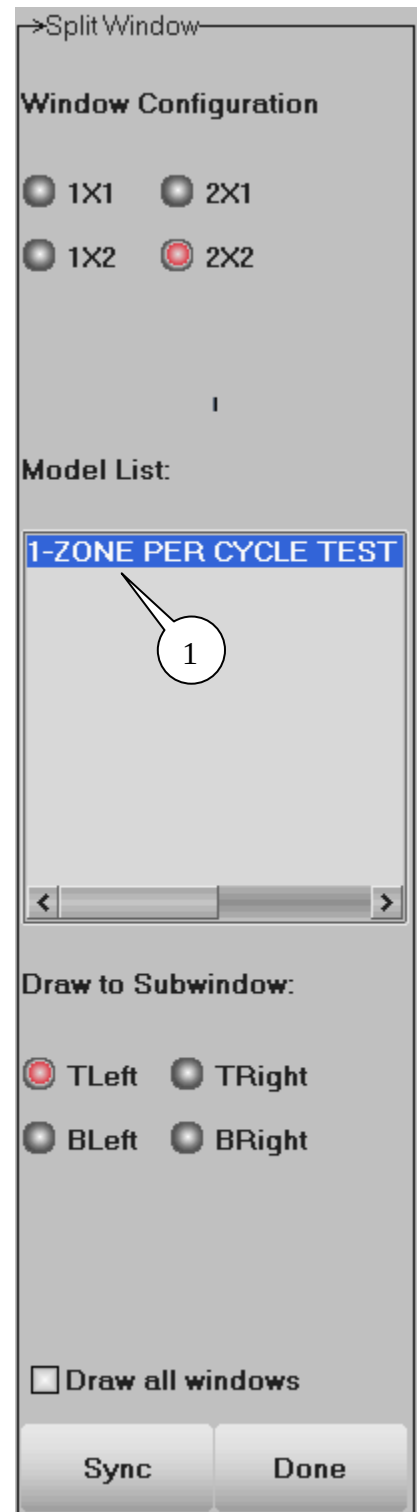
Quick Guides

Quick Guide: How to use multiple windows

1. Select number of drawing areas
2. Select active drawing area using TLeft...BRight
3. Select model to view in active drawing area
(Use File/Open to open more models)

Each active drawing area can be operated upon independently unless Draw to all windows is used.

Use right click on drawing area to make it active





NODE TRACE

Introduction

Use this interface to trace the paths of nodes over time.

Interface Description

Pick Node – Pick nodes to trace

Area – Select nodes inside/outside a user defined area

Polygon – Select nodes inside/outside a user defined region

In – Select nodes inside the area/polygon

Out – Select nodes outside the area/polygon

Key in Node IDs – Enter node IDs to trace

Node label – Turn node label on/off

Line Width – Select trace line width

Line Color – Select trace line color

Starting State – Enter starting state number

[Write Curve] – Write trace curves for selected nodes

[Write Trace] – Write coordinate history of selected nodes

[Clear Trace] – Clear traced entities

[Done] – Exit Node Trace interface

Quick Guides

Quick Guide: How to generate a node trace

1. Select trace line width/color
2. Pick nodes to trace using single pick/area/polygon
3. Animate model

Quick Guide: How to write a node trace curve

1. Select trace line width/color
2. Enter starting state
3. Pick nodes to trace using single pick/area/polygon
4. Click [Write Curve]

Quick Guide: How to write a node trace in text format

1. Select trace line width/color
2. Enter starting state
3. Pick nodes to trace using single pick/area/polygon
4. Click [Write Trace]

The screenshot shows the 'Node Trace' interface with the following elements:

- Pick Node**: A red circle icon.
- Area**: A radio button.
- Polygon**: A radio button.
- In**: A radio button.
- Out**: A radio button.
- Key in Node IDs:**: A text input field.
- Node Label**: A checkbox with a checked mark.
- Line Width**: A dropdown menu showing '1'.
- Line Color**: A dropdown menu showing 'White'.
- Starting State:**: A text input field showing '1'.
- Write Curve**: A button.
- Write Trace**: A button.
- Clear Trace**: A button.
- Done**: A button.



LIGHTING SETUP

Introduction

Use this interface to apply different lighting effects using up to ten independently setup lights.

Interface Description

Select light number to define light

On – Turn highlighted light on

Off – Turn highlighted light off

Ambient – Adjust the ambient component of light

Diffuse – Adjust the diffuse component of light

Specular – Adjust the specular component of light

[Reset] – Reset light to default values

RGB – Set RGB values

X, Y, Z-pos – Enter X, Y, Z coordinates for light source MP

Quick Guides

Quick Guide: How to setup model lighting

1. Select a light
2. Turn it on
3. Select Ambient and adjust sliders
4. Select Diffuse and adjust sliders
5. Select Specular and adjust sliders
6. Enter coordinates for light source (Hit enter to accept each value)

The screenshot shows the 'Lighting Setup' window. At the top, there's a title bar 'Lighting Setup'. Below it, on the left, are two radio buttons: 'On' (selected) and 'Off'. To the right of these is a list of lights from 'Light0' to 'Light9'. 'Light0' is highlighted. Below the list, there are three radio buttons for 'Ambient' (selected), 'Diffuse', and 'Specular'. To the right of these is a 'Reset' button. Below the radio buttons are three sliders for 'R', 'G', and 'B' (representing RGB values), each with a blue star icon and a value of 30. At the bottom, there are three input fields for 'X-pos', 'Y-pos', and 'Z-pos' with values 0.0, 0.0, and 1.0 respectively.

Light	On/Off	Ambient	Diffuse	Specular	R	G	B	X-pos	Y-pos	Z-pos
Light0	On	Selected			30	30	30	0.0	0.0	1.0
Light1	Off									
Light2	Off									
Light3	Off									
Light4	Off									
Light5	Off									
Light6	Off									
Light7	Off									
Light8	Off									
Light9	Off									



SET DISPLAY OPTIONS

Introduction

Use this interface to organize personal display preferences.

Interface Description

Axes/Surface – Open element axes and shell surface integration points interface

Displ. factor – Open displacement scale factor interface

Reflections – Open global reflections interface

Hic.Csi const. – Open Hic/Csi constants interface

SPH nodes – Open SPH display options interface

Thickness Scale Factor – Open shell thickness scale factor interface

FLD E'Strain – Toggle FLD Strain (Engineering/True)

Local B'Strain – Toggle Local Brick Strain (General Electric)

El Axes – Select element axes for stress results

Surface – Select position in shells for stress results

Select shell integration point

(Only applicable when Surface = IntPt / BeamIPt)

[Aply] – Apply selected options to the model

[Clr] – Clear and picked parts from the list

[Rset] – Reset model to default options

[Done] – Exit Set Display Options interface

Set Display Options

☒ Axes/Surface

☐ Disp. scale factor

☐ Reflections

☐ Hic/Csi const.

☐ SPH nodes

☐ Thickness Scale Factor

☐ Fringe Scale Factor

☒ FLD E'Strain

☐ Local B'Strain

El Axes: Global

Surface: Maxima

Aply Clr Rset Done



Displacement Scale Factor

Whole – Displace whole model

Part – Displace part of model

Displacement Scale Factor – Enter/Select displacement scale factor

X, Y, Z – Toggle directions to apply displacement scale factor

Whole ☒ Part ☐

Displacement Scale:

1.0 1.0

☒ X ☒ Y ☒ Z

Reflections

Whole – Reflect whole model

Part – Reflect part of model

Reflect about – Toggle planes to reflect about

Whole ☒ Part ☐

Reflect about:

☐ XY ☐ YZ ☐ XZ

Head Injury Criteria (Hic) / Chest Severity Index (Csi)

Time Units – Enter/Select Hic time units

Gravity constant – Enter/Select Hic gravity constant

Time units:

1 sec

Gravity constant:

9.81 9.81



SPH nodes

SPH Radius Scale – Enter/Select SPH radius scale factor

SPH Sphere divs – Enter/Select SPH sphere divisions

Style – Select SPH sphere style

Fixed radius – Keep SPH radius constant

SPH nodes control panel showing input fields for Sph Radius Scale (0.3), Sph Sphere divs (12), Style (point), and a checkbox for Fixed Radius.

Shell Thickness Scale Factor

Thickness Scale Factor – Enter/Select shell thickness scale factor

Change in Thickness Scale Factor – Enter/Select change in shell thickness scale factor

Shell Thickness Scale Factor control panel showing radio buttons for Whole and Part, and input fields for Thickness Scale Factor (1.0) and Change in Thickness Scale (1.0).

Fringe Scale Factor

Fringe Scale Factor – Select/Enter fringe scale factor



Vector Plot

Introduction

Display normal vectors for any element in the model.

Interface Description

Select Vector Type

X, Y, Z – Toggle X,Y,Z components of vector

Vector range – Enter minimum and maximum values for vector range
(Press enter to accept values)

Dynamic – A set of min/max ranges is computed for each time state

Static – A constant min/max range is computed using all time states

User – Range set by user, enter min/max values above

Show – Shows elements within the range entered above

SF – Enter scale factor for vector plot

Hidden line vector off – Switch off hidden line for vectors

(If a vector is behind part of the model it will not be shown, check this option to display the vector)

Whole – Apply vector plot to whole model

Part – Pick parts for vector plot

Area – Define an area for vector plot

El/Node – Pick an element or node for vector plot

[Apply] – Apply vector plot

[Clear] – Clear vector plot

[Done] – Exit Vector Plot interface

Quick Guides

Quick Guide: How to display a vector plot

1. Select a vector type
2. Select a direction (if applicable)
3. Select vector range type
(Enter Min/Max values for User and Show)
4. Select/Enter vector arrow scale factor
5. Select entities
6. Click [Apply] to display vector plot

The screenshot shows the 'Vector Plot' dialog box. At the top, 'Velocity' is selected in a dropdown menu. Below it are three checkboxes for 'X', 'Y', and 'Z' components, all of which are checked. The 'Vector Range' section contains two input fields for 'Min' and 'Max', both set to '0'. There are four radio buttons: 'Dynamic' (selected), 'Static', 'User', and 'Show'. The 'SF' (Scale Factor) section has two input fields, both set to '1.0'. Below this is a checkbox for 'Hidden line vector off', which is currently unchecked. At the bottom, there are four radio buttons for entity selection: 'Whole' (selected), 'Part', 'Area', and 'El/Node'. At the very bottom are three buttons: 'Apply', 'Clear', and 'Done'.



IDENTIFY NODE/ELEMENT/PART

Introduction

Use this interface to display the ID for any node/element/part in the model.

Interface Description

Pick Entity – Pick and identify an entity

Area – Identify entities inside/outside a user defined area

Polygon – Identify entities inside/outside a user defined region

In – Identify entities inside the area/polygon

Out – Identify entities outside the area/polygon

Key in ID – Enter entity IDs to be highlighted

Key in xyz coord – Enter XYZ coordinates to be highlighted
(Separate each value by a space/coma)

Node – Identify nodes

Element – Identify elements

Part – Identify parts

Show Results – Show Fringe Component results in Command Window

[Unpick last] – Unpick last picked entity

[Clear Node] – Clear all identified nodes

[Clear Ele] – Clear all identified elements

[Clear Part] – Clear all identified parts

[AllVis] – Identify all visible entities

[Clear All] – Clear all identified entities

[Done] – Exit Identify Node/Element/Part interface

Quick Guides

Quick Guide: How to identify nodes/elements/parts

1. Select entity type to identify

2. Select picking method and pick entities from model

Or enter entity ID / x, y, z coordinates

Information will be displayed in Command Window

(Note to use Show Results first apply a Fringe Component to the model using the Fcomp interface)



TIME HISTORY RESULTS

Introduction

Use this interface to display and plot data for various data over time.

Interface Description

Global – Select global history plot

Nodal – Select nodal history plot

Element – Select element history plot

Int.pt. – Select element integration point history plot

Material – Select material history plot

R-Nodal – Select relative nodal history plot

Scalar – Select fringed scalar history plot

Vol-fail – Select volume of material failure option

Sum mats – sum materials for material history plot

Select a time history component (1)

Value – Select element values or max/min element value for material

Elm – Plot data for selected elements

Max – Plot data for element with max value for selected material

Min – Plot data for element with min value for selected material

E-type – Select element type for time history plotting

E-axes – Select shell element axes for time history plotting

Surface – Select shell stress surface position

[Plot] – Plot time history data in current XY-Plot window

[New] – Plot time history data in a new XY-Plot window

[Padd] – Add time history data to current XY-Plot window

[Raise] – Raise all open XY-Plot windows

[Pop] – Open and Raise all closed XY-Plot windows

[Clear] – Clear selected items in list

[Done] – Exit Time History Plot interface

Time History Results

☐ Global ☐ Material

☐ Nodal ☐ R-Nodal

☒ Element ☐ Scalar

☐ Int.pt. ☐ Vol-fail

☐ Sum mats

X-stress
Y-stress
Z-stress
XY-stress
YZ-stress
ZX-stress
Effective Plastic Strain
Pressure
Effective Stress (v-m)
Max Prin Dev Stress
2nd Prin Dev Stress
Min Prin Dev Stress
Maximum Shear Stress

Value: Elm

E-Type: Any

E-Axes: Global

Surface: Maxima

Plot New Padd

Clear Raise Pop Done



Vol-Fail: Volume failure

Failure levels – Select failure levels from list

Eroded – Add item to failure list

[Add] – Add entered value to the failure levels list

[Clr] – Clear text field and selected items

[Del] – Delete selected item from the failure levels list

Value – Enter failure level value to be added to the list

Quick Guides

Quick Guide: How to display a general time history plot

1. Select a time history type
2. Select time history component
3. Pick nodes for Nodal and R-Nodal/elements for Element and Int. Pt./materials for Materials/select a fringe component for Scalar (Use Fcomp)
4. Click [Plot]/[New]

Quick Guide: How to display an element time history plot

1. Select Element
2. Select element type
3. Select time history component
4. Select element value type (Elm/Max/Min)
5. Select element axes Global/Local
6. Select shell surface stress position
7. Pick elements from model
8. Click [Plot]/[New]

Quick Guide: Using Volume Failure plotting with 1 level and n materials

1. Enter Failure level
2. Click [Add]
3. Pick n materials from drawing area
4. Click [Plot] / [New]

Quick Guide: Using Volume Failure plotting with n levels and 1 material

1. Enter Failure level
2. Click [Add] (repeat to enter multiple levels)
3. Pick 1 material from drawing area
4. Click [Plot] / [New]



SET PART COLOR

Introduction

Use this interface to apply different colors and transparency levels to selected parts. This information can be saved using the Save/Retrieve Views interface.

Interface Description

Pick Part – Assign color/transparency by part

Area – Assign color/transparency to parts inside/outside a user defined area

Polygon – Assign color/transparency to parts inside/outside a user defined region

In – Assign color/transparency to parts inside the area/polygon

Out – Assign color/transparency to parts outside the area/polygon

Color – Set color for parts

Transp – Set transparency for parts

Part Colormap – Color map used to assign colors to parts automatically, can be used as a pallet to assign colors to parts manually

Editmap – Clicking on the color map will assign a chosen color to the color map (toggle on/off)

[Reset] – Reset the color map to default settings

[AllVis] – Assign color/transparency to all visible parts

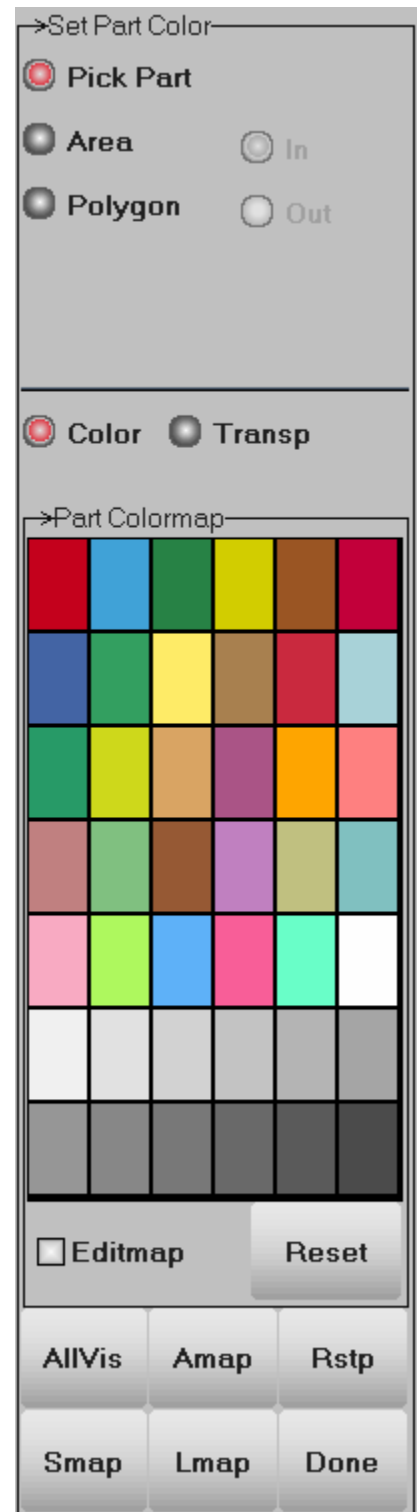
[Amap] – Apply the current color map to parts

[Rstp] – Reset transparency of all parts to opaque

[Smap] – Save part colors to an ASCII file

[Lmap] – Load part colors from an ASCII file

[Done] – Exit Set Part Color interface





Color Options

1

RGB – Use sliders/text fields to set values for each color component

Color Chooser – Displays a user defined color or a color picked from a part/color map. (1)

Set – Assign user defined color to part/color map/option

Show – Show picked part/source color in the color chooser

Options

[Backg] – Apply selected color as plain background color

[Mesh] – Apply selected color as mesh color

[Hilite] – Apply selected color as highlight color

[Text] – Apply selected color as text color

[Label] – Apply selected color as label color

[Outlin] – Apply selected color as outline color

[Sky] – Apply selected color as sky color, (for Fade, Dia-fade and Tri-fade backgrounds)

[Ground] – Apply selected color as ground color, (for Fade, Dia-fade and Tri-fade backgrounds)

[Middle] – Apply selected color as middle color, (for Tri-fade background)

Transparency Options

Transparency Factor – Transparency Factor component slider

Set Transparency – Assign transparency to a part

Get Transparency – Show part transparency value on slider



Quick Guides

Quick Guide: How to assign a color to a part/option

1. Select Color
 2. Select show
 3. Select a color from the color map/generate a color using the sliders
 4. Select set
 5. Select parts using pick/area/polygon
- Or select an option from the lower interface

Quick Guide: How to assign a color to the color map

1. Select Color
2. Select a color from the color map/generate a color using the sliders
3. Check Editmap
4. Select box in color map to assign new color

Quick Guide: How to assign a transparency to a part

1. Select Transp
 2. Select Set Transparency
 3. Move slider to desired transparency
- (Note: The slider will not set a transparency of 0, use Rstp)
4. Select parts using pick/area/polygon



ELEMENT BLANKING

Introduction

Use this interface to blank selected elements.

Interface Description

Pick Element – Blank a single element

Area – Blank elements inside/outside a user defined area

Polygon - Blank elements inside/outside a user defined region

Sphere – Blank elements inside/outside a spherical region

Unblank Part – Pick an element to un-blank the whole part

In – Blank elements inside the area/polygon

Out – Blank elements outside the area/polygon

Center – Enter a node ID or x, y, z coordinates to define the center of the sphere

Radius – Enter radius for spherical blanking

Set Radius – Use slider to set radius for spherical blanking

[Blank Sphere] – Press to perform spherical blank

[Blank All] – Blank all visible elements

[UnBlank All] – Un-blank all elements

[Reverse] – Reverse blanked/un-blanked elements

[Update Surf] – Update the 3D outer surface

[UpdEdge] – Update edge line

[UnBlank Last] – UnBlank last blanked elements

[Done] – Exit Element Blanking interface

Auto Update – Auto update the 3D outer surface

Surface Only – Blank surface of solid elements only

Quick Guides

Quick Guide: How to blank elements by using an area/polygon

1. Select area/polygon
2. Select In/Out
3. Define area/region
(Note: Elements will be blanked automatically)

Quick Guide: How to blank elements using a sphere

1. Select Sphere
2. Pick a node from the model to be the origin
3. Enter a radius/move slider
4. Click blank sphere



EXPLODE PARTS

Introduction

Use this interface to explode/separate parts so that they can be viewed more easily.

Interface Description

Factor – Enter/Select an explode scale factor

Directions – Select directions to explode model

Move – Move exploded parts using mouse pointer

All – Move /explode all parts

Part – Move/explode picked parts only

PtGroup – Move/explode part groups only

Part Groups List – Select part group from list

[Apply] – Explode selected parts

[Reset] – Reset model

[Clear] – Clear picked parts and list selections

[Done] – Exit Explode Parts interface

Quick Guides

Quick Guide: How to explode a set of parts

1. Set scale factor
2. Set direction
3. Select parts to explode
4. Click Apply

Quick Guide: Using Move

1. Toggle Move to activate
2. Left mouse button operates rotation
3. Middle mouse button translates along axis
4. Right mouse button translates 90deg to axis indicated in 3.

→Explode Parts

Factor:

1.0 1.0

Directions: xyz

☐ Move

☐ All

☐ Part

☒ PtGroup

Part Groups list:

Apply Clear

Reset Done



CROSS PLOTTING

Introduction

Control all open XY-Plot windows and files using this interface.

Interface Description

File – Show list of XY-Plot data files

Window – Show list of current XY-Plot windows

Select file/window to be shown (1)

Select XY-Plot to show or to cross plot (2)

[Add] – Open and add a XY-Plot data file to the filename list

[Rem] – Remove a XY-Plot data file from the filename list

[Info] – Show full XY-Plot data file path in command window

Show – Show selected plot

Cross – Cross selected plots

[Plot] – Plot XY-Plot data in current XY-Plot window

[New] – Plot XY-Plot data in a new XY-Plot window

[Padd] – Add XY-Plot data to current XY-Plot window

[Clear] – Clear selected items in list

[Raise] – Raise all open XY-Plot windows

[Pop] – Open and Raise all closed XY-Plot windows

[Done] – Exit Cross Plotting interface

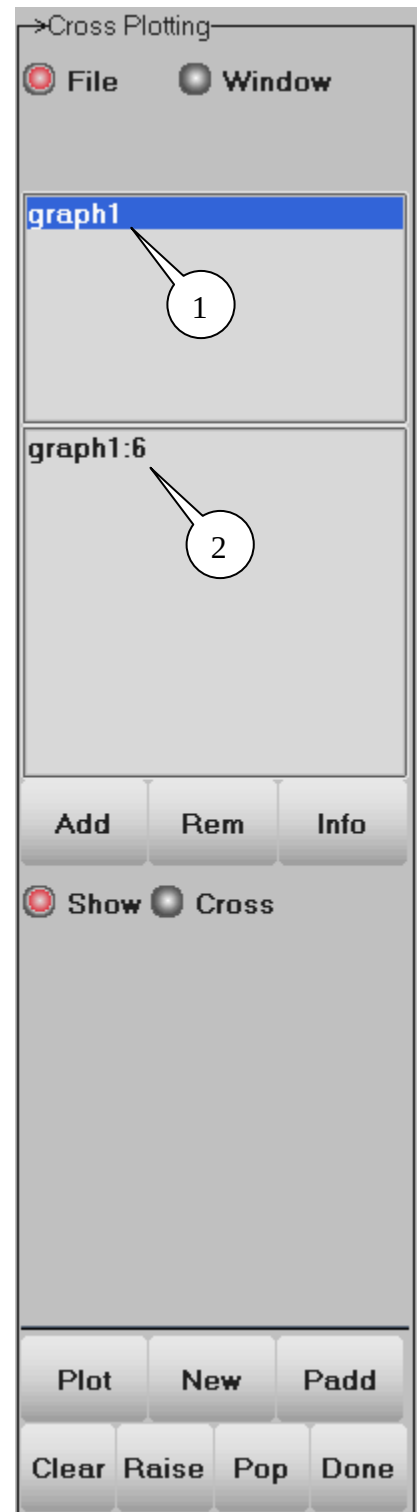
Quick Guides

Quick Guide: Cross Plotting – File

1. Select File
2. Click Add to open a new XY-Plot data file
3. Select the new file
4. Select multiple stored XY-graphs
5. Select show and click [Plot]/[New] to open
6. Select cross and click [Plot]/[New] to cross graphs

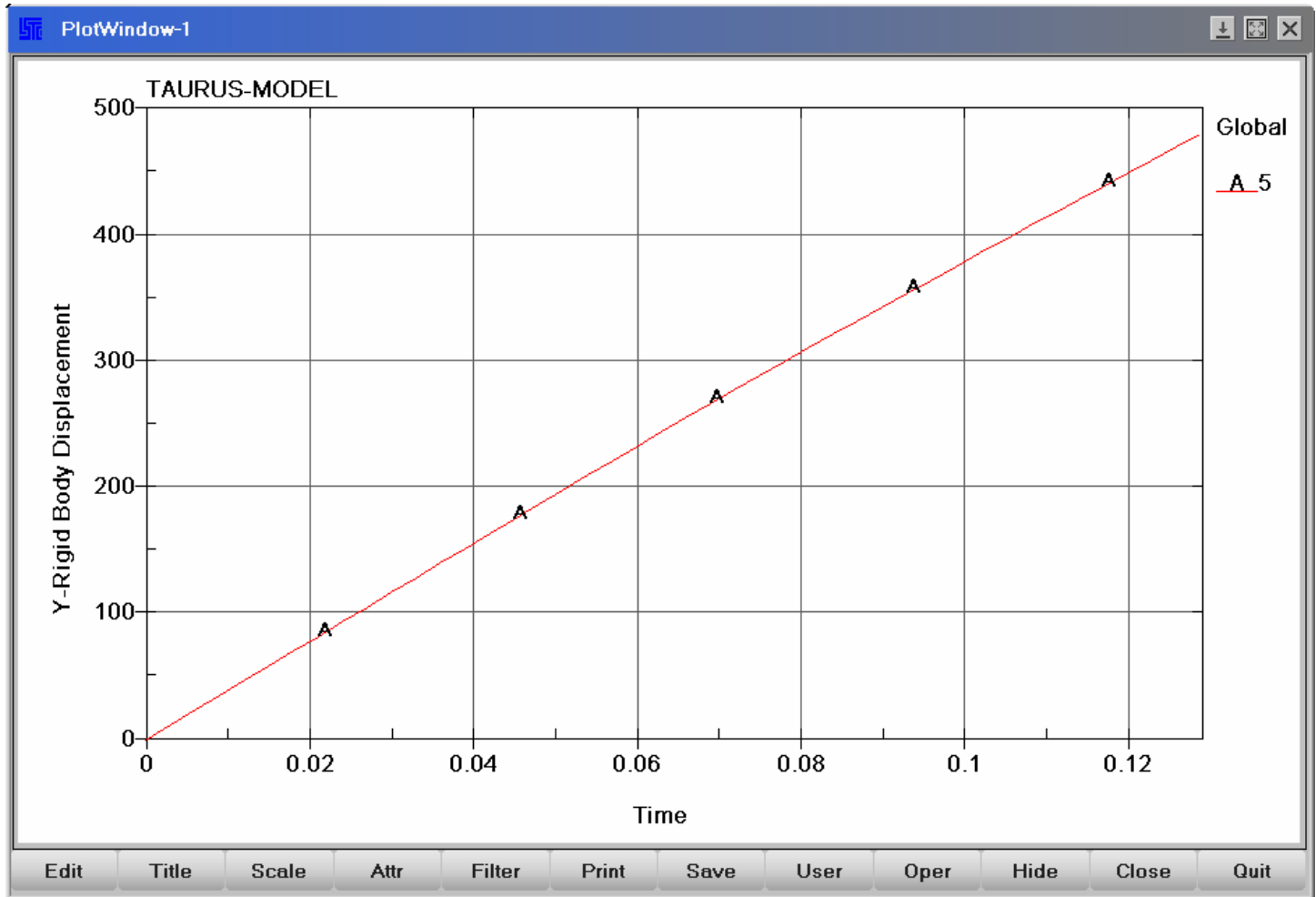
Quick Guide: Cross Plotting – Window

1. Select Window
2. Select window from the main list
3. Select a curve that is in that window
4. Click [New] to open curve in a new window





XY-PLOT GRAPHICS WINDOWS



[Edit] – Edit XY-Plot Data (Not implemented yet)

[Title] – Set and change titles/axis labels

[Scale] – Set and change offset/scale of curves

[Attr] – Change attributes of XY-Plot

[Filter] – Set curve filtering

[Print] – Open print dialog

[Save] – Write XY-Plot data to a file

[User] – Load XY-Plot data from a file

[Oper] – Process XY-Plot data

[Hide] – Hide buttons (Esc to restore them)

[Close] – Close the plot window

[Quit] – Close and delete plot window from the program memory

Left Menu

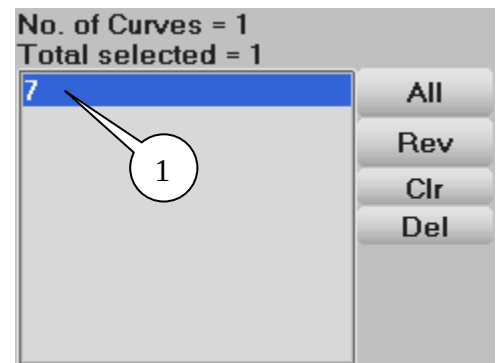
[All] – Select all curves in the list

[Rev] – Reverse curve selection

[Clr] – Clear curve selection

[Del] – Remove selected curve from list

Curve List (1)





Right Menu

Grid – Add/remove grid lines in XY-Plot window
Tick – Add/remove tick marks in XY-Plot window
Frame – Add/remove frame in XY-Plot window
Legend – Add/remove legend in XY-Plot window
Autofit – Toggle Automatically Fit data in XY-Plot window (on/off)
Timeline – Toggle time line in XY-Plot window
Invert – Invert background color in XY-Plot window
Maxgraph – Use all of XY-Plot window to display data
MinMax – Add/remove min/max label in XY-Plot window

☒	Grid
☒	Tick
☒	Frame
☒	Legend
☒	Autofit
☐	Timeline
☐	Invert
☐	Maxgraph
☐	Minmax

Title

All Rev Clr Del	Plot Title: TAURUS-MODEL				
	X-Axis Label: Time	Y-Axis Label: Y-Rigid Body Displacement			
	☐ Mod Curve Legend	Legend Label: Global			
Show All		Show Select	Apply	Reset	Done

Plot Title – Input plot title
X/Y-Axis labels – Input X/Y axis labels
Modify Curve Legend – Modify curve name in legend (Press enter to update)
Legend Title – Input legend title
[Show All] – Show all curves in XY-Plot window

[Show Select] – Show selected curves in XY-Plot window
[Apply] – Apply settings to XY-Plot
[Reset] – Reset settings to default values
[Done] – Close the XY-Plot options menu

Scale

All Rev Clr Del	Xmin	0	Xmax	0.128998	Axes: Lin-Lin
	Ymin	0	Ymax	480.636	
	X-offset	0	X-scale	1	
	Y-offset	0	Y-scale	1	
Show All		Show Select	Apply	Reset	Done

Xmin/max – Input min/max values for X-axis.
Ymin/max – Input min/max values for Y-axis.
X-offset – Input offset value for X-axis.
Axes – Select logarithmic/linear scale for each axis.

Y-offset – Input offset value for Y-axis.
X-scale – X axis scale factor
Y-scale – Y axis scale factor



Attributes

All Rev Clr Del	Pts/sym	4	Symbol	—	▼	Apply	Reset
	☐ Join symbols with line		Color	■	▼	Apply	Reset
	Auto Symbol		Width	—	▼	Apply	Reset
			Style	—	▼	Apply	Reset
Show All		Show Select		Apply		Reset	Done

Pts/Sym – Input number of points per symbol

Join symbols with line – Add/remove lines between symbols when using auto symbol

[Auto Symbol] – Automatically replace curve with symbols

Symbol – Select symbol style (Use [Apply]/[Reset] to make individual changes)

Color – Select curve and symbol color (Use [Apply]/[Reset] to make individual changes)

Width – Select line width (Use [Apply]/[Reset] to make individual changes)

Style – Select line style (Use [Apply]/[Reset] to make individual changes)

Quick Guides

Quick Guide: Setting XY-Graph attributes using Auto Symbol

1. Select curves
2. Enter the number of points per symbol
3. Enable/Disable Join symbols with line
4. Click [Auto Symbol] this will add symbols to more than 1 line

Quick Guide: Setting XY-Graph attributes manually

1. Select a single curve
2. Select Symbol type, click [Apply] to apply symbol type only
3. Select line color, click [Apply] to apply line color only
4. Select line width, click [Apply] to apply line width only
5. Select style, click [Apply] to apply style only
6. Click [Apply] at bottom to apply all changes



Filter

All Rev Clr Del	Filter	none	Time	msec	C/s(Hz)	60	
	Point Average	none					
Show All		Show Select		Apply		Reset	Done

Filter – Select filter to apply

Time – Select time units

C/s(Hz) – Select/Enter frequency

Point Average – Select number of points to average.

Ex, PA=3, then an average of the ordinate values is taken every 3 points and each of the 3 points is then given this new value.

Quick Guides

Quick Guide: How to apply a filter

1. Select a curve
2. Select filter type
3. Select time units
4. Select frequency
5. Select point average
6. Click [Apply] to apply all changes

Print

(See Print Dialog information under drop down menus)



Save

The Save dialog box features a vertical toolbar on the left with buttons: All, Rev, Clr, and Del. The main area contains 'Output Type:' with a dropdown menu set to 'Curve file', and 'Output Interval:' with a dropdown menu set to '1' and a text input field containing '1'. Below these is the 'Enter Filename:' label and a large text input field. At the bottom are five buttons: Show All, Show Select, Save, Reset, and Done.

Output Type – Select data output type

Output Interval – Select/Enter number of time steps between outputs

Filename - Enter filename including full path with no spaces

[Save] – Write XY-Plot data to given filename

Quick Guides

Quick Guide: Saving XY-Plot data to a file

1. Select curves to output
2. Select output format
3. Select output interval
4. Enter filename (If no path is given file will be saved in the same directory as the executable)
5. Click [Save]

Load

The Load dialog box features a vertical toolbar on the left with buttons: All, Rev, Clr, and Del. The main area contains a 'Filename list:' label above a large empty list box. To the right of the list box are two buttons: 'Add File' and 'Remove File'. At the bottom are five buttons: Show All, Show Select, Select, Reset, and Done.

Filename list – Select a file to show stored data.

[Add File] – Add a file to the filename list.

[Remove File] – Remove a file from the filename list.

[Select] – Open selected data files.



Quick Guides

Quick Guide: How to load a XY-Plot data file

1. Click [Add File]
2. Select file and click [Open]
3. Select file in filename list
4. Click [Select]
5. Stored curves should now be visible in the left menu, select curves from here to display them



Operations

The screenshot shows a software interface for post-processing operations. On the left, a vertical list contains the following operations: differentiate, integrate, sum_curves, subtract_curves, multiply_curves, divide_curves, invertx, and inverty. To the right of this list are three checkboxes labeled 'Curve1:', 'Curve2:', and 'Curve3:', each followed by a text input field. To the right of these is a 'Time units:' section with a text input field containing '1' and a dropdown menu showing 'sec'. Below that is a 'Gravity constant:' section with a text input field containing '9.81' and a dropdown menu showing '9.81'. At the bottom of the main area is a 'Save Result:' label followed by a text input field. At the very bottom of the window are five buttons: 'Show All', 'Show Select', 'Apply', 'Reset', and 'Done'.

Operations List – Select an operation to apply to a curve

Curve 1, 2, and 3 – Pick/Enter curves from curve list for operation

Time Units – Select/Enter HIC time units

Gravity Constants – Select/Enter HIC gravity constants

[Save Result:] – Save resultant curve as a XY-Plot data file

Quick Guides

Quick Guide: Operating on a curve

1. Select operation from the list
2. Select a curve(s) from the left list
3. Click [Apply]

Quick Guide: How to subtract/multiply/divide curves

1. Select appropriate operation
2. Select/Enter curve 1 from left list
3. Select/Enter curve 2 from left list
4. Click [Apply]

Quick Guide: How to apply resultant2 and 3

1. Select appropriate operation
2. Select/Enter curve 1 from left list
3. Select/Enter curve 2 from left list
4. Select/Enter curve 3 from left list (resultant 3 only)
5. Click [Apply]

Quick Guide: How to apply Hic15/Hic36/CSI

1. Select appropriate operation
2. Select/Enter time
3. Select/Enter gravitation constant
4. Click [Apply]

Quick Guide: Saving results



1. Carry out operations as described above
2. Enter a new filename in the text field provided
3. Click [Save Result]



FORMING LIMIT RESULTS

Introduction

Use this interface for metal forming analysis.

Interface Description

t – Enter sheet thickness in mm

n – Enter FLD crit. formula index

[Set] – Apply the new t and n values

t_n – Set thickness and index of FLD curve

File – Read thickness and index of FLD curve from file

FLD Data File list – Select FLD data file,
(use File/Open/FLD Curve to open a new file) (1)

Point – Select an item for point on FLD plot

Tracer – Select an item for tracer on FLD plot

Material – Select a material for FLD plot

Element – Select an element for FLD plot

Area – Define an area for FLD plot.

Polygon – Define a region for FLD plot.

In – Select entities within the area/polygon.

Out – Select entities outside the area/polygon.

[Clr] – Clear information in popup windows.

[Info] – Open/close FLD information dialog.

[FormaDia] – Open/close formability diagram dialog.

Strain results for last element picked (2)

Position – Select shell surface for FLD results

[Plot] – Plot FLD diagram in current XY-Plot window

[New] – Plot FLD diagram data in a new XY-Plot window

[Padd] – Add data to FLD diagram in current XY-Plot window

[Clear] – Clear picked entities

[Raise] – Raise all open XY-Plot windows

[Pop] – Open and Raise all closed XY-Plot windows

[Done] – Exit Forming Limit Result interface

Forming Limit Results

Calculated FLD Curve:

t= 0.8 n= 0.21 Set

☒ t_n ☐ File

1

☒ Point ☐ Tracer

☐ Material ☐ Element ☒ Area ☐ Polygon

Clr Info

FormaDia

☒ In ☐ Out

El#=1284, Con=1398,1423,1

T=0, %R=0
E1=0, E2=0

2

Position: Middle

Plot New Padd

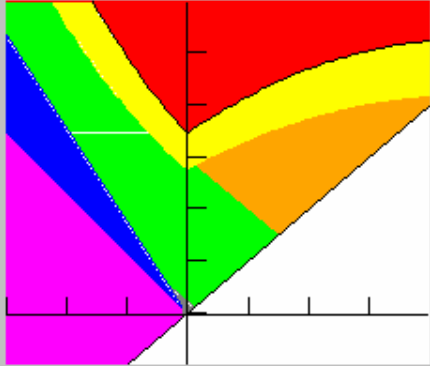
Clear Raise Pop Done



Formability Diagram

 Formability Diagram - CRLCS (t=0.8 n=0.21)

Limit % FLC:	0	☒ Cracks
Safety margin % FLC:	20	☒ Risk of cracks
Allowable thinning:	0.3	☒ Severe thinning
Essential thinning:	0.02	☒ Inadequate stretch
Allowable thickening:	0.01	
R-value:	1.82	☒ Wrinkling tendency
		☒ Wrinkles



Close

Limit % FLC – Enter % limit of Forming Limit Curve

Safety margin % FLC – Enter safety margin percentage from Forming Limit Curve

Allowable thinning – Enter level of acceptable thinning

Essential thinning – Enter required amount of thinning

Allowable thickening – Enter acceptable amount of thickening

R-value – Enter material R-value

Cracks – Show crack region

Risk of cracks – Show risk of cracks region

Severe thinning – Show excessive thinning region

Inadequate stretch – Show lack of stretch region

Wrinkling tendency – Show tendency to wrinkle region

Wrinkles – Show wrinkled region

[Close] – Close Formability Diagram dialog

Quick Guides

Quick Guide: Displaying Forming Limit Diagram

1. Enter sheet thickness and FLD criterion (t and n)
Or select File and open a pre loaded FLD curve file
(Use File/Open/FLD curve to load a FLD curve file)
2. Click [FormaDia] to set FLD parameters
3. Select a material/element using single or area pick
4. Click [Plot]/[New] to display results



STATE TIME

Introduction

Use this interface to activate/deactivate time states and apply overlays to the model.

Interface Description

State List (1)

Choose – Selecting a state from the state list will display it

Delete – Selecting a state from the state list will make it inactive

Turn overlay mode on/off (2)

Overlay State – Enter state number for overlay

Overlay Color – Select overlay color

Overlay Type – Select overlay type

[Reset] – Reset all states to active

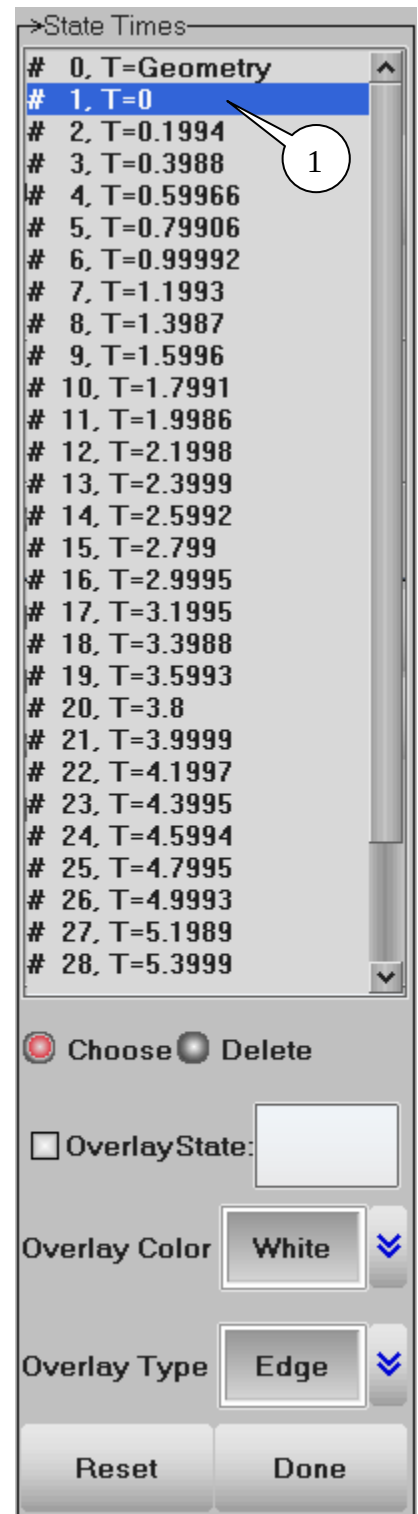
[Done] – Exit State Time interface

Quick Guides

Quick Guide: How to use overlays

1. Click on (2) to activate overlay mode
2. Select overlay color
3. Select overlay type
4. Select a state from the list

(Now when using measure differences between the overlay and current state will also be displayed in the command window)





MEASURE

Introduction

Use this interface to make measurements of various aspects of the model and to create local coordinate systems.

Interface Description

Active Element Only – Measure part using active elements only

Item – Select an item to measure/Create local coordinate system

[Cancel pick] – Cancel last picked node (For Angle 3/4Node / Create Axes)

Element – Display measurement for selected elements

Part – Display measurement for selected parts

All – Display measurement for all visible parts

[Apply] -

[Delete] – Delete selected local axes

Reference Axes – Select current reference axes for measurement (1)

History – Select history type to plot (2)

[Plot] – Plot History data in current XY-Plot window

[New] – Plot History data in a new XY-Plot window

[Padd] – Add History data to current XY-Plot window

[Clear] – Clear selected items in list

[Raise] – Raise all open XY-Plot windows

[Pop] – Open and Raise all closed XY-Plot windows

[Done] – Exit Measure interface

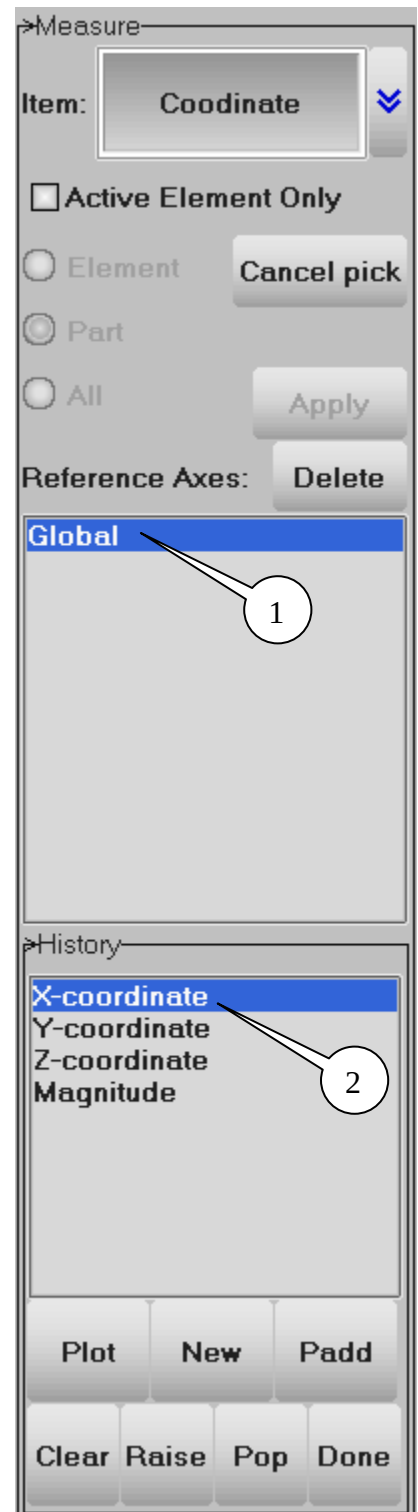
Quick Guides

Quick Guide: How to measure a coordinate

1. Select Reference Axes
2. Select Coordinate
3. Pick 1 Node
4. Coordinate for current state is displayed in command window (lower left)
5. Select history type
6. Click [Plot]/[New]/[PAdd] to view time history plot

Quick Guide: How to measure a distance

1. Select Reference Axes
2. Select Item: Distance / Distance N2S
3. Pick 2 nodes / 1 node and 1 element
4. Distance for current state is displayed in command window (lower left)
5. Select history type
6. Click [Plot]/[New]/[PAdd] to view time history plot





Quick Guide: How to measure an angle

1. Select Reference Axes
2. Select Angle 3Nodes / Angle 4Nodes
3. Pick 3 nodes / Pick 4 nodes, 2 per line
4. Angle for current state is displayed in command window (lower left)
5. Select history type
6. Click [Plot]/[New]/[PAdd] to view time history plot

Quick Guide: How to measure an area

1. Select Reference Axes
2. Select Item: Area
3. Select Element/Part/All (Click [Apply] if using All)
4. Check Active Elements Only to ignore blanked elements
(Part/Element/All, area for current state is displayed in command window)
5. Select history type
6. Click [Plot]/[New]/[PAdd] to view time history plot

Quick Guide: How to measure a volume/mass

1. Select Reference Axes
2. Select Item: Volume/Mass
3. Select Element/Part/All (Click [Apply] if using All)
(Part/Element/All, volume/mass for current state is displayed in command window)
4. Select history type
5. Click [Plot]/[New] to view time history plot

Quick Guide: How to measure inertia/angular velocity

1. Select Reference Axes
2. Select Item: Inertia/Ang Velocity
3. Select Part/All (Click [Apply] if using All)
(Part/All, Inertia/Angular velocity for current state displayed in command window)
4. Select history type
5. Click [Plot]/[New]/[PAdd] to view time history plot

Quick Guide: How to define a local coordinate system

1. Select Item: Create Axes
 2. Pick 3 nodes to generate local coordinate system
- To delete,
1. Select local coordinate system to remove
 2. Click [Delete]



ASCII

Introduction

Use this interface to browse and display data contained in various ASCII data files output by DYNA.

Interface Description

File – Load an ASCII file from an alternative directory

Load – Load existing file for selected ASCII file type

Uload – Un-Load selected ASCII file (to free memory)

[Raise] – Raise all open XY-Plot windows

[Pop] – Open and Raise all closed XY-Plot windows

[Done] – Exit ASCII interface

[Plot] – Plot items from ASCII file in current XY-Plot window

[New] – Plot items from ASCII file in a new XY-Plot window

[Padd] – Add items from ASCII file to current XY-Plot window

[All] – Select all ASCII items

[Clear] – Clear all selections

[Rev] – Reverse selection

[Info] – Show information on the loaded ASCII file

ASCII items list – select items loaded from ASCII file(1)

Main Branch(1)

Sub Branch(2)

{matsum}[Total] – Toggle the combining of multiple ASCII items

Affects {secforc}, {spcforc}, {nodfor}, {bndout}, {sleout} and {gceout}

{abstat}[Airbag/Material] – Select airbag/material IDs

{elout}[Clrcp] – Clear Elout component list

{elout}[Stress/Strain/Beams] – Select Elout component types

{deforc}[Trans/Rotation] – Select translational force /rotational moment of spring element

{spcforc}[Force] – Select forces for plotting

{spcforc}[Momt] – Select moment for plotting

{jntforc}[Jforc] – Select joint forces for plotting

{jntforc}[Jstif] – Select joint stiffness for plotting

{rbdout}[Local] – Plot curve in local coordinate system

{sphout}[Stress/Strain] – Select sphout component types

{nodout}[HicCsi] – Open Hic/Csi options interface





Head Injury Criteria (Hic) / Chest Severity Index (Csi) Options

Hic Csi Options Pre-Filter options

T: 1 sec Filter none

G: 9.81 1.81 Pt Ave none

C/sHz: 60 60

Done

T – Enter/Select time units for Hic/Csi/Filter

Filter – Select filter type for filtering data

G – Enter/Select gravitation constant in model units

Pt Ave – Select number of points for point averaging filter

C/sHz – Enter/Select filtering frequency in model time units

[Done] – Exit Hic/Csi Options interface

Quick Guides

Quick Guide: How to load an ASCII item

1. Select ASCII item to load from list
2. Click [Load]
3. Find ASCII file for that item
4. Click [Open]

Quick Guide: How to plot stored data

1. Load ASCII item
2. Select Main branch
3. Select IDs (If applicable)
4. Select Sub-branch
5. Click [Plot]/[New]/[PAdd]

Quick Guide: How to set Hic/Csi options

1. Load {nodout} ASCII item
2. Click [Hic/Csi]
3. Enter Time for Hic
4. Enter Gravitation Constant for Csi
5. Select filter type
6. Select number of points per average
7. Enter/Select frequency
8. Click done to close interface



SAVE/RETRIEVE VIEWS

Introduction

Use this interface to save and retrieve information regarding model appearance, color and orientation.

Interface Description

List of saved views(1)

View Name_number – Enter a new view name

Color – Toggle Save/Retrieve model color on/off

Appearance – Toggle Save/Retrieve model appearance on/off

Orientation – Toggle Save/Retrieve model orientation on/off

[Save] – Save the current view to memory

[Retrieve] – Retrieve selected view from memory

[Delete] – Delete selected view from memory

[Done] – Exit Save/Retrieve Views interface

Quick Guides

Quick Guide: How to save a view

1. Enter view name
2. Select what to save (Color/Appearance/Orientation)
3. Click save

Quick Guide: How to retrieve a view

1. Select view from list
2. Select what to retrieve (Color/Appearance/Orientation)
3. Click Retrieve

Quick Guide: How to delete a view

1. Select view from list
2. Click delete

Save/Retrieve Views

View_01

1

View Name_number:

View_02

☒ Color

☒ Appearance

☒ Orientation

Save Retrieve

Delete Done



MODEL SELECTION

Introduction

Use this interface to open and select multiple models.

Interface Description

Model List – Select model to display (1)

[Delete] – Delete the highlighted model from program memory

[Reset] – Reset model to original position

[Done] – Exit Model Selection interface

[X] – Select direction for model translation

Translation Distance – Enter distance to translate model

Multi Model Selection – Activate multiple models (Note once activated model selection for translation etc can only be done from the Model ID list and not the Model List. The Model List is used to select models for multiple selections)

Sync States – Synchronize animation for multiple models

Translation – Enter amount for translation

Model ID List – Select model for translation when using multi model selection

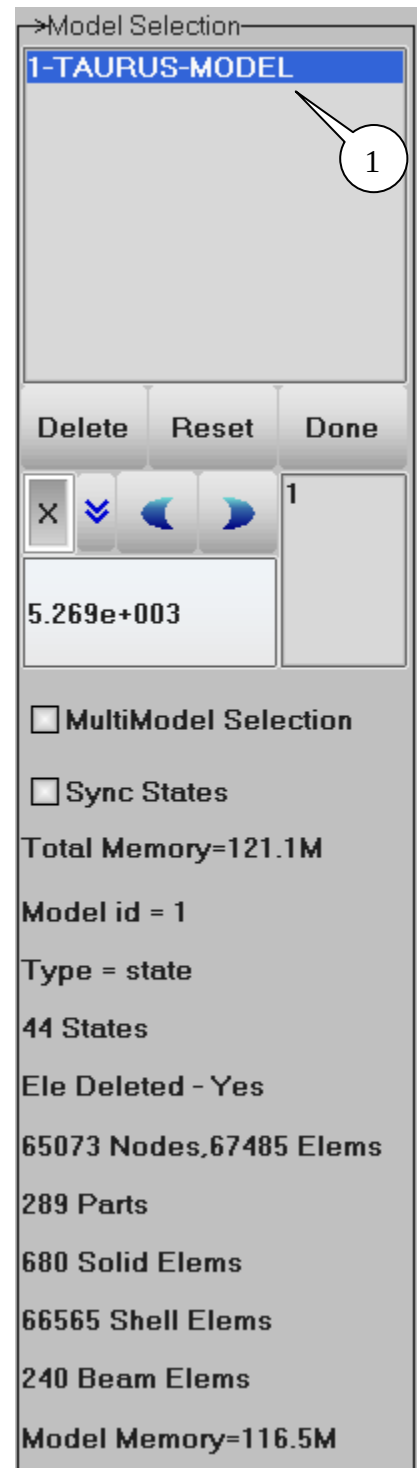
Quick Guides

Quick Guide: Using multiple model selection

1. Open models using File/Open
2. Activate multiple model selection
3. Select models to be displayed together from the Model list

Quick Guide: Solving overlapping models

1. Select one of the over lapping models(1)
2. Select direction of translation
3. Click left or right arrow to translate model





PART SELECTION

Introduction

Use this interface to select which parts to display.

Interface Description

Beam – Select/deselect Beam elements

Shell – Select/deselect Shell elements

Solid – Select/deselect Solid elements

Tshell – Select/deselect Tshell elements

Mass – Select/deselect Mass elements

Discrete – Select/deselect Discrete elements

Seatbelt – Select/deselect Seatbelt elements

Inertia – Select/deselect Inertia elements

Rsurf – Select/deselect Rigid-surface elements

SPHnd – Select/deselect SPH nodes elements

Fluid – Select/deselect Fluid elements

Single – Remove/Keep parts using single pick

Area – Remove/Keep parts inside a user defined area

Polygon – Remove/Keep parts inside a user defined region

[Save] – Save current active parts to selected buffer

[Load] – Load parts from selected buffer

[Buff(n)] – Select part buffer to save/load

Rm –Remove parts inside the area/polygon

Kp –Keep parts inside the area/polygon

[Info] – Show information on selected parts

[All] – Select all parts.

[None] – Deselect all parts

[Rev] – Reverse part selection

[Auto] – Update drawing with selection automatically, (Red=on).

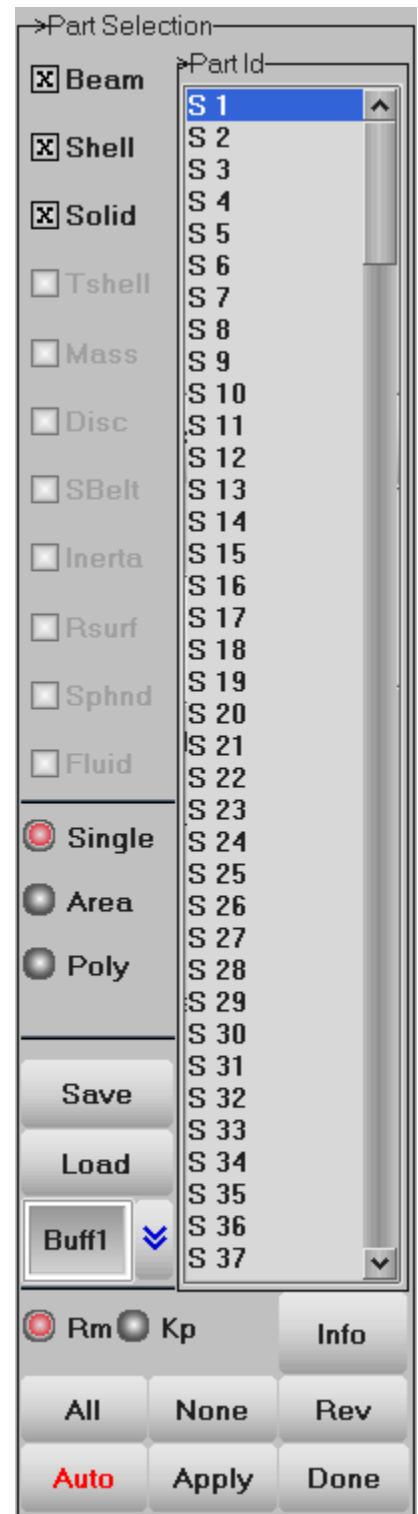
[Apply] – Update drawing with selected parts. (when Auto=off).

[Done] – Exit Part Selection interface

Quick Guides

Quick Guide: How to use the Part Selection Interface

1. Select Remove or Keep
2. Select parts from the list
Or pick from the model
3. Part types can be selected by checking the appropriate box on the left side
4. If Auto is not active (red) click [Apply]





PAGE 2:PRE/POST PROCESSING

INTRODUCTION

The first four interfaces are for post processing, while the others are for pre processing. The pre processing capabilities include model operations like transform, translate, reflect, rotate, project and scale. Soon to be implemented will be element creation and error checking.

CFD	Cgat	
Binout	Skid	
EIEdit	Dupgrid	
Movcopy	Elcheck	Normals
Trnsfrm	Reflect	Project
Translt	Rotate	Scale



CFD TIME HISTORY PLOT

Introduction

Interface Description

Node list for history plot (1)

History Comp – List of Time History Components (2)

[Load] – Load D3THINS file to plot

[Uload] – Un-Load D3THINS files to free memory

[Plot] – Plot XY-Plot data in current XY-Plot window

[New] – Plot XY-Plot data in a new XY-Plot window

[Padd] – Add XY-Plot data to current XY-Plot window

[Clear] – Clear selected nodes

[Raise] – Raise all open XY-Plot windows

[Pop] – Open and Raise all closed XY-Plot windows

[Done] – Exit CFD Time History Plot interface

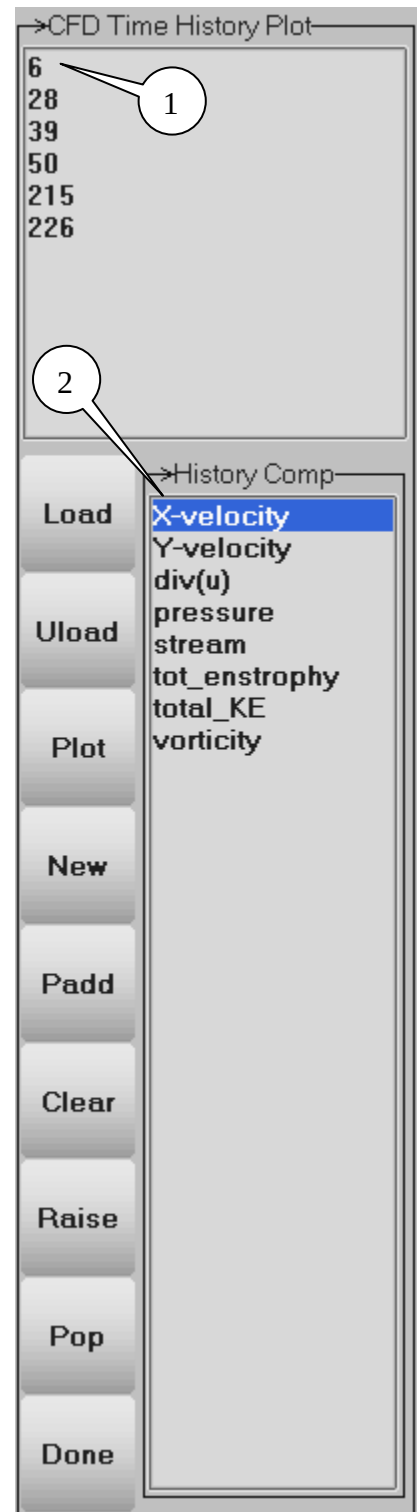
Quick Guides

Quick Guide: How to load a CFD time history component

1. Click [Load]
2. Select d3thins or d3mean file and click [Open]
(Node and component data should load)

Quick Guide: How to view a CFD time history plot

1. Select node(s) from list (1)
2. Select History Component(2)
3. Click [Plot]/[New]





BINOUT

Introduction

Use this interface to browse and display data stored in binary ASCII files. The branch system makes it easy to navigate and there is even the option to open multiple files for comparison.

Interface Description

[Load] – Load a BINOUT file

[UnLoad] – Un-load selected BINOUT file

[Save] – Save selected BINOUT branch to a file

[Done] – Exit BINOUT interface

Open Files list – Select from list of currently loaded BINOUT files (1)

Main Branch list – Select main branches in selected BINOUT file (2)

Sub-Branch list – Select sub-branch part of selected main branch (3)

List of main branch entities (4)

List of main branch components (5)

[Plot] – Plot selected component in current XY-Plot window

[New] – Plot selected component in a new XY-Plot window

[Padd] – Add selected component to current XY-Plot window

[Pop] – Open and Raise all closed XY-Plot windows

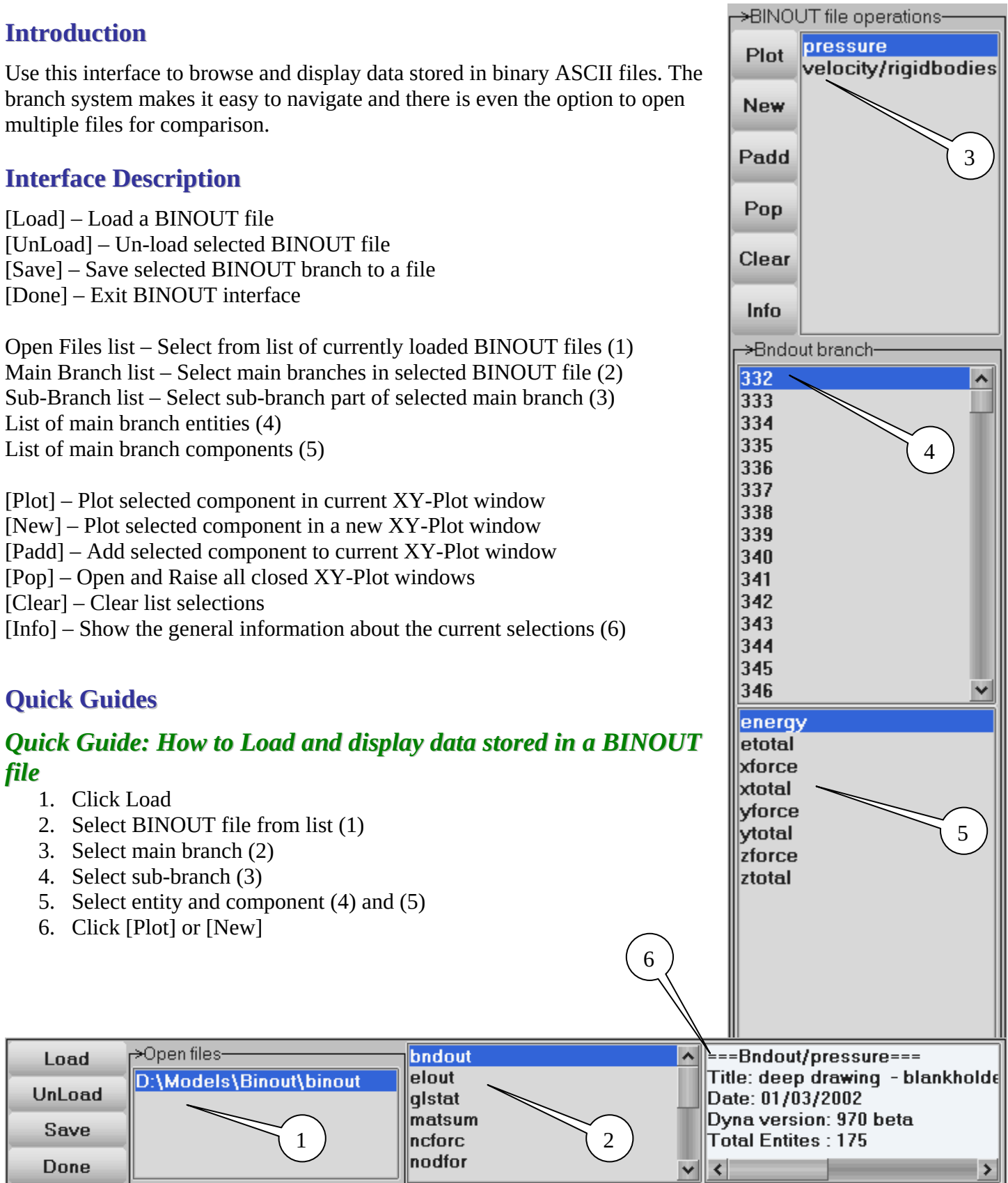
[Clear] – Clear list selections

[Info] – Show the general information about the current selections (6)

Quick Guides

Quick Guide: How to Load and display data stored in a BINOUT file

1. Click Load
2. Select BINOUT file from list (1)
3. Select main branch (2)
4. Select sub-branch (3)
5. Select entity and component (4) and (5)
6. Click [Plot] or [New]





CGAT – CIRCULAR GRID ANALYSIS TECHNIQUE

Introduction

Interface Description

Circles X– Number of circles in local X direction

Circles Y– Number of circles in local Y direction

Diameter – Diameter of circles

Spacing – Distance between circles

Origin – Pick/Enter origin of circular grid (X,Y,Z coordinates)

Xpoint – Pick/Enter direction of grid x-axes (X,Y,Z coordinates)

Y-node – Pick/Enter a node for y-axes

Part No. – Enter part number or part pick at origin

Color – Color of circle line

Linew – Line width of circles

Pick – Pick/Enter circle ID e.g.: 2x3 for history plot

[Show] – Generate and display Grid circles

[Hide] – Remove grid circles from display

[Clear] – Delete all generated grid circles

[Print] – Print size and % distortion at current state

[Plot] – Plot history of pick circles

[New] – Plot history of pick circles in a new window

[Padd] – Add plot history of pick circles current plot

[Done] – Exit Circular Grid interface

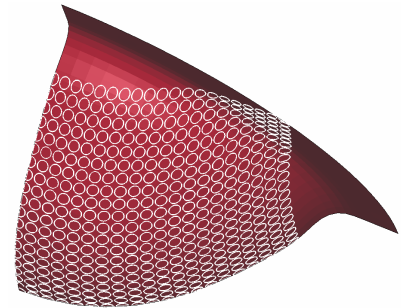
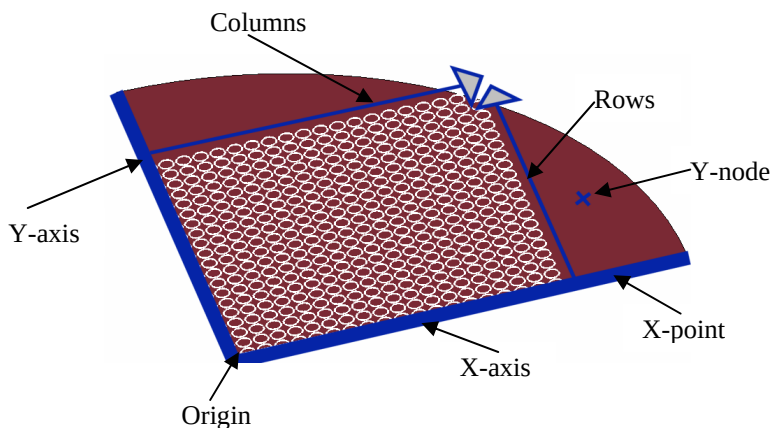
→Cgat

Circles X	Y
20	20
Diameter	Spacing
2.04545	2.45455
☒ Origin	☐ Xpoint
0.0	0.0
0.0	0.0
0.0	0.0
☐ Y-node	
Part No	1
Color	White
Linew	1
☐ Pick	1x1
Show	Hide
Clear	Print
Plot	New
Padd	Done

Quick Guides

Quick Guide: How to form a grid of circles

1. Enter number of columns (circles in X-direction)
2. Enter number of rows (circles in the Y-direction)
3. Enter circle diameter (calculated by default per model)
4. Enter spacing, measured from origin to origin
5. Check Origin to pick origin of grid
Or Enter XYZ coordinates in text fields
6. Check Xpoint to pick point to form X-axis of grid
Or Enter XYZ coordinates in text fields
7. Check Y-node to pick a point to from a XY plane for the grid
Or Enter node ID in text field
8. Enter Part ID of part to apply circles
9. Select circle line color
10. Select Circle line width
11. Click [Show]



Quick Guide: How to view time history data for a circle

1. Check Pick to pick a circle from the grid
2. Or Enter the circle's grid location e.g.: 2x3
3. Click [Plot]/[New]/[Padd]



SKID

Introduction

Interface Description

Pick blank – Pick blank part for skid tracing
Pick curve – Pick points along tool for skid tracing
Pick point – Pick a point on tool for skid tracing
Closest node – Displays node nearest picked coordinates

[Clr] – Clear all points in curve list
[Del] – Delete highlighted point from curve list
[Rep] – Replace highlighted point in curve list
[Add] – Add new point after highlighted point in curve list

Curve List: – List of curve coordinates

Skid – Change skid trace color/width
Tool – Change tool line color/width
Start – Change start line color/width
End – Change end line color/width
[Color] – Select line color
[Linew] – Select line width
Pick trace – Pick/Enter skid trace for time history plot

[Proj] – Project picked points/curve onto tool
[Show] – Generate and display skid mark tracks
[Hide] – Remove skid marks from display
[Clear] – Delete all generated skid marks
[Plot] – Plot history of pick skid mark
[New] – Plot history of pick skid mark in a new window
[Padd] – Add plot history of pick skid mark current plot
[Done] – Exit Skid Mark interface

The screenshot shows the 'Skid marks' interface with the following elements:

- Skid marks** header with a right-pointing arrow.
- ☒ **Pick blank** with a text input field.
- ☐ **Pick curve** and ☐ **Pick point**.
- X:** text input field.
- Y:** text input field.
- Z:** text input field.
- Closest node** with a text input field.
- Buttons: **Clr**, **Del**, **Rep**, **Add**.
- Curve Coords:** a large empty rectangular area.
- Set line color/width** section:
 - ☒ **Skid** and ☐ **Tool**.
 - ☐ **Begin** and ☐ **End**.
 - A color selection dropdown showing **White** and a width selection dropdown showing **2**.
- ☐ **Pick trace** with a text input field containing **1**.
- Buttons: **Proj**, **Show**, **Hide**, **Clear**, **Plot**, **New**, **Padd**, **Done**.



Quick Guides

Quick Guide: How to display a skid

1. Place model at state 1
 2. Check Pick blank to pick the blank part
Or Enter Part ID in text field
 3. Check Pick curve to pick multiple points on the tool
Or Check Pick Point to pick a single point on the tool (click [Add] to add point to curve list)
 4. Select Skid/Tool/Begin/End
 5. Select line color
 6. Select line width
 7. Click [Proj]
- Increment states to display the skid marks

Quick Guide: How to view time history data for a skid trace

1. Check Pick trace to pick a skid trace
Or Enter the trace number
2. Click [Plot]/[New]/[Padd]



GENERAL SELECTION MENU

☒ Pick	☐ In	Keyin		☐ Label	Selecting Nodes			
☐ Area	☐ Out	LabOn	Save	Whole	ByNode	BySet		
☐ Poly	☐ Add	LabOff	Load	All Vis	ByElem			
☒ Rm		Clear	Desel	Rev	ByPart	3Dsurf		

Pick – Select/Deselect a single entity

Area – Select/Deselect entities inside/outside a user defined area

Poly – Select/Deselect entities inside/outside a user defined region

In – Select/Deselect entities inside the area/polygon

Out – Select/Deselect entities outside the area/polygon

Add – Add entities to selection

Rm – Remove entities from selection

[Keyin] – Press this button to enter entity IDs (Hit enter to accept value)

Label – Turn label on/off for new selection

LabOn – Display labels for all entities

LabOff – Hide labels for all entities

Clear – Clear all entity selections

Save – Save entities to buffer

Load – Load saved entities from buffer

Desel – Deselect last selected entity

Whole – Select all entities within model

Allvis – Select all visible entities

Rev – Reverse entity selection

By Node – Selection by node

By Elem – Selection by element

By Part – Selection by part

By Set – Select entities using SET data

3Dsurf – Toggle select surface nodes/elements on/off (3D parts only)

Quick Guides

Quick Guide: Selecting nodes/elements/parts

1. Select picking method Pick/Area/Poly
 2. Select In/Out for area/poly
 3. Select Add/Remove
 4. Select picking 'by' method (Note 'by SET' opens a link dialog)
 5. Make selections from model
- Or Enter node/element/part ID by clicking [Keyin] (Hit enter to accept entry)



MOVE OR COPY ELEMENTS

Introduction

Use this interface to change element Part IDs.

Interface Description

Move – Move elements to a target part

Copy – Copy elements to a target part

[Apply] – Apply move or copy operation

Pick Target Part – Pick target part from drawing area

Part ID – Enter target part ID

Element ID Offset – New element IDs will be created using the existing IDs + ID value

Starting Element ID + 1 – New element IDs will be created by incrementing the ID value.

Element ID Offset – Enter ID value, used as ID offset/starting ID

Node ID Offset – Enter node ID offset

[Done] – Exit Move/Copy interface

Quick Guides

Quick Guide: How to change an element's association with a part

1. Select elements using the general selection interface
2. Check Pick Target Part to pick from a part ID from the model
Or Enter a Part ID (Hit enter to accept the value)
(This is the Part ID that will be applied to the selected elements)
3. Click [Apply]

Quick Guide: How to copy an element and apply an association with a part

1. Select elements using the general selection interface
2. Check Pick Target Part to pick from a part ID from the model
Or Enter a Part ID (Hit enter to accept the value)
(This is the Part ID that will be applied to the selected elements)
3. Select Elem ID Offset to use an offset to create the new element IDs, enter the offset value in the box labeled ID (Hit enter to accept the value)
Or Select Starting Elem ID + 1 to use a new set of IDs to create the new element IDs. Enter the starting ID in the box labeled ID (Hit enter to accept the value)
4. Enter a node ID offset to create the new node IDs
5. Click [Apply]

→Move or copy Elements

☐ Move

☒ Copy

Apply

☐ Pick Target Part

Part ID:

☒ ELem ID Offset

☐ Starting ELem ID +

ID: 8805

Node ID Offset:

Done



TRANSFORM MODEL

Introduction

Use this interface to transform parts of the model.

Interface Description

[Transform] – Apply transformation

From – Form the coordinate system the model will transform from

To – Form the coordinate system the model will transform to

Node1F, 2F and 3F – Pick or enter node IDs to form the coordinate system the model will transform from

Node1T, 2T and 3T – Pick or enter node IDs to form the coordinate system the model will transform to

Copy elements – Copy elements while transforming the model

Transfer – Transfer selection to newly copied elements

Offset – EID Offset is the offset for the new element ID

Start ID – EID Offset is the starting element ID

Pick Part – Pick target part from drawing area or enter part ID

[Plist] – Open part list dialog

EID Offset – Enter element ID offset or starting element ID

NID Offset – Enter node ID offset

[Undo] – Undo last operation

[Done] – Exit Transform Model interface

Quick Guides

Quick Guide: How to transform a model

1. Select nodes/elements/parts to transform
2. Check From
3. Select nodes to form From coordinate system
4. Check To
5. Select nodes to form To coordinate system
6. Click [Transform]

Quick Guide: How to copy and transform a model

1. Follow above guide up to step 5
2. Select and enter a new starting ID
Or Select and enter an ID offset
3. To associate new elements to a part pick/select/enter a part ID
4. Click [Transform]



TRANSLATE MODEL

Introduction

Use this interface to translate entities.

Interface Description

[Tran -] – Translate in negative direction

[Tran +] – Translate in positive direction

Direction – Select direction of translation

Translate distance – Enter distance to translate

Node 1, 2 and 3 – Pick/Enter node IDs to define N1-N2, N1-N2-N3 and N1 to N2 directions

(See Transform Interface for details on Copy Elements interface)

[Undo] – Undo last operation

[Done] – Exit Translate Model interface

Quick Guides

Quick Guide: How to Translate a model

1. Select nodes by node/element/part using pick/area/region
2. Select Direction
3. If using N1-N2/N1-N2-N3/N1 to N2 then define node IDs (note when using N1 to N2 the distance between the N1 and N2 will be the distance used for translation)
4. Click [Tran - / +] to translate entities

Quick Guide: How to Translate a model

1. Follow above guide up to step 6
2. Check Copy Elements
3. Select and enter a new starting ID
Or Select and enter an ID offset
4. To associate new elements to a part pick/select/enter a part ID
5. Click [Tran - / +] to translate nodes/elements/parts

→Translate Model

Tran - Tran +

Direction GLOBAL X

Translate distance:

☐ Node1 ☐ Node2 ☐ Node3

☒ Copy Elements

☒ Offset ☐ StartId

☐ PickPart Plist

Eid Offset 8805

Nid Offset 218

Undo Done



REFLECT MODEL

Introduction

Use this interface to reflect the model.

Interface Description

[Reflect] – Perform reflection for the selected nodes

Pick node as pt. of reflec – Pick a node as the origin of the reflection

Node ID – Enter a node ID as the origin of the reflection (Hit enter to accept)

XYZ – Enter x,y,z coordinates as the origin of the reflection

Plane – Select plane of reflection

Node 1, 2 and 3 – Enter node IDs to define planes using N1-N2 and N1-N2-N3

(See Transform Interface for details on Copy Elements interface)

[Undo] – Undo last operation

[Done] – Exit Reflect Model interface

Quick Guides

Quick Guide: How to reflect a model

1. Select nodes by node/element/part using pick/area/region
2. Pick/Enter a node ID or x, y, z coordinates as the origin of the reflection
3. Select a plane of reflection
4. If using N1-N2/N1-N2-N3 then define node IDs
5. Click [Reflect]

Quick Guide: How to copy and reflect a model

1. Follow above guide up to step 4
2. Check Copy Elements
3. Select and enter a new starting ID
Or Select and enter an ID offset
4. To associate new elements to a part pick/select/enter a part ID
5. Click [Reflect]

The screenshot shows the 'Reflect Model' dialog box. At the top is a 'Reflect' button. Below it is a checkbox 'Pick node as pt. of reflec'. There are three input fields: 'Nodeid:', 'XYZ:', and 'Plane:'. The 'Plane:' dropdown is currently set to 'Norm X'. Below these are three checkboxes labeled 'Node1', 'Node2', and 'Node3', each with an adjacent input field. A section titled 'Copy Elements' has a checked checkbox. Below this are two radio buttons: 'Offset' (selected) and 'StartId'. There is a 'PickPart' checkbox with an adjacent input field and a 'Plist' button. Below these are two more input fields: 'Eid Offset' with the value '8805' and 'Nid Offset' with the value '218'. At the bottom are 'Undo' and 'Done' buttons.



ROTATE MODEL

Introduction

Use this interface to rotate the model.

Interface Description

Rotate -/+ – Rotate the selected nodes in negative / positive direction

Pick node as origin – Pick a node as the origin of rotation

Node ID – Enter node ID as the origin of rotation (Hit enter to accept)

XYZ – Enter x, y, z coordinates as the origin of rotation

Rot. Axis – Select axis/plane of rotation

Rot. Angle – Enter angle of rotation in degrees

Node1, 2 and 3 – Enter node IDs for N1-N2 and N1-N2-N3 planes

(See Transform Interface for details on Copy Elements interface)

[Undo] – Undo last operation

[Done] – Exit Rotate Model interface

Quick Guides

Quick Guide: How to rotate a model

1. Select nodes by node/element/part using pick/area/region
2. Pick/Enter a node ID or x, y, z coordinates as the origin of rotation
3. Select an axis/plane of rotation
4. If using N1-N2/N1-N2-N3 then define node IDs
5. Enter rotation angle
6. Click [Rotate]

Quick Guide: How to copy and rotate a model

1. Follow above guide up to step 5
2. Check Copy Elements
3. Select and enter a new starting ID
Or Select and enter an ID offset
4. To associate new elements to a part pick/select/enter a part ID
5. Click [Rotate]

Rotate Model

Rotate - Rotate +

☐ Pick node as origin

NodeID:

XYZ:

Rot. Axis: Global X

Rot. Angle:

☐ Node1

☐ Node2

☐ Node3

☒ Copy Elements

☒ Offset ☐ StartId

☐ PickPart

Eid Offset

Nid Offset

Undo Done



PROJECT MODEL

Introduction

Use this interface to create a projection of the model.

Interface Description

[Project] – Perform projection for selected nodes

Pick node on plane of p – Pick a node on the plane of projection

Node ID – Enter a node ID on the plane of projection (Hit enter to accept)

XYZ – Enter x, y, z coordinates on the plane of projection

Plane – Select plane of projection

Node 1, 2 and 3 – Enter node IDs for N1-N2 and N1-N2-N3 planes

(See Transform Interface for details on Copy Elements interface)

[Undo] – Undo last operation

[Done] – Exit Project Model interface

Quick Guides

Quick Guide: How to project a model

1. Select nodes by node/element/part using pick/area/region
2. Pick/Enter a node ID or x, y, z coordinates for a point on the plane of projection
3. Select a plane of projection
4. If using N1-N2/N1-N2-N3 then define node IDs
5. Click [Project]

Quick Guide: How to copy and project a model

1. Follow above guide up to step 4
2. Check Copy Elements
3. Select and enter a new starting ID
Or Select and enter an ID offset
4. To associate new elements to a part pick/select/enter a part ID
5. Click [Project]

Project Model

Project

☐ Pick node on plane of p

Nodeid:

XYZ:

Plane:

☐ Node1

☐ Node2

☐ Node3

☐ Copy Elements

☐ Offset ☐ StartId

☐ PickPar

Eid Offset

Nid Offset



SCALE MODEL

Introduction

Use this interface to alter the scale of the model.

Interface Description

[Scale Down/Up] – Scale the model down/up with selected nodes

Pick node as origin – Pick a node as the origin of scaling

Node ID – Enter a node ID as the origin of scaling (Hit enter to accept)

XYZ – Enter x, y, z coordinates as the origin of scaling

Scale Dir – Select direction of scaling

Scale Factor – Enter scale factor

Node 1, 2 and 3 – Enter node IDs for N1-N2 and N1-N2-N3 planes

(See Transform Interface for details on Copy Elements interface)

[Undo] – Undo last operation

[Done] – Exit Scale Model interface

Quick Guides

Quick Guide: How to scale a model

1. Select nodes by node/element/part using pick/area/region
2. Pick/Enter a node ID or x, y, z coordinates as the origin of scaling
3. Select a direction/plane of scaling
4. If using N1-N2/N1-N2-N3 then define node IDs
5. Enter scale factor
6. Click [Scale Down/Up]

Quick Guide: How to copy and scale a model

1. Follow above guide up to step 4
2. Check Copy Elements
3. Select and enter a new starting ID
Or Select and enter an ID offset
4. To associate new elements to a part pick/select/enter a part ID
5. Click [Scale Down/Up]

Scale Model

Scale Down Scale Up

☐ Pick node as origin

NodeID:

XYZ:

Scale Dir: Global X

Scale Factor:

☐ Node1

☐ Node2

☐ Node3

☐ Copy Elements

☐ Offset ☐ StartId

☐ PickPar

Eid Offset

Nid Offset

Undo Done



SHELL/SEGMENT/TSHELL NORMALS

Introduction

Use this interface to display shell/segment/tshell normals

Interface Description

Select entity type(1)

Show Normal – Click element to Show element normal

Reverse Normal – Interface to reverse element normal

Vector size – Set normal vector size (>1.0) larger

Pick element – Pick element to reverse normal

Pick seed – Pick elements as a seed for auto-reverse

[Autoreverse] – Auto-reverse normal according to seed element

[Done] – Exit Shell/Segment/Tshell Normals interface

Quick Guides

Quick Guide: How to display Shell/Segment/Tshell normals

1. Select normal type
2. Select Show/Reverse normal





PAGE 3: KEYWORD FILE EDITING

Introduction

Page 3 provides viewing and editing tools for LS-DYNA keyword files. Each main keyword has its own section accessible via the buttons shown below. Each section contains a list of sub-keywords which can be selected opening individual keyword input dialogs. Although page 3 allows the creation of individual keyword entries, to create a keyword file from scratch use the tools on Page 4.

	*Damp	*Load
*Airbag	*Dbase	*Mat
*Ale	*Define	*Node
*Boundry	*Element	*Part
*Cnstrnd	*Eos	*Rgdwall
*Compnt	*Hrglass	*Section
*Contact	*Initial	*Set
*Control	*Intgrtn	*Termnt
*Deform	*Intrfac	*User



KEYWORD SELECTION MENU

Access the main keyword branch by selecting from the main menu shown on the previous page. This will open up main branch navigation interfaces below, in the dynamic interface area. To open a specific keyword select it from this list and click [Edit]. There are additional options available for specific keywords and these are described below.

General Keyword Options

[Edit] – Open keyword input window for selected keyword

[All] – Display all keywords

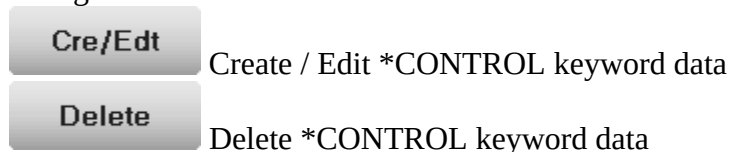
[Model] – Display all keywords in current model

[Done] – Close the current interface



*CONTROL Extra Keyword Options

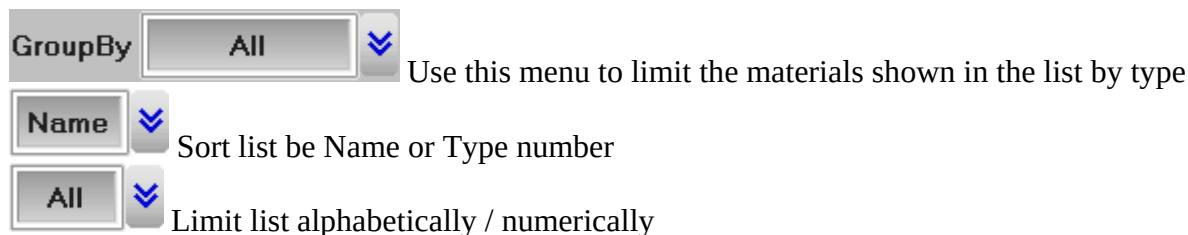
As there can only be one of each *CONTROL keyword type they are deleted here instead of the keyword input dialog.



*EOS / *HOURLASS / *MAT Extra Keyword Options



*MAT Extra Keyword Options

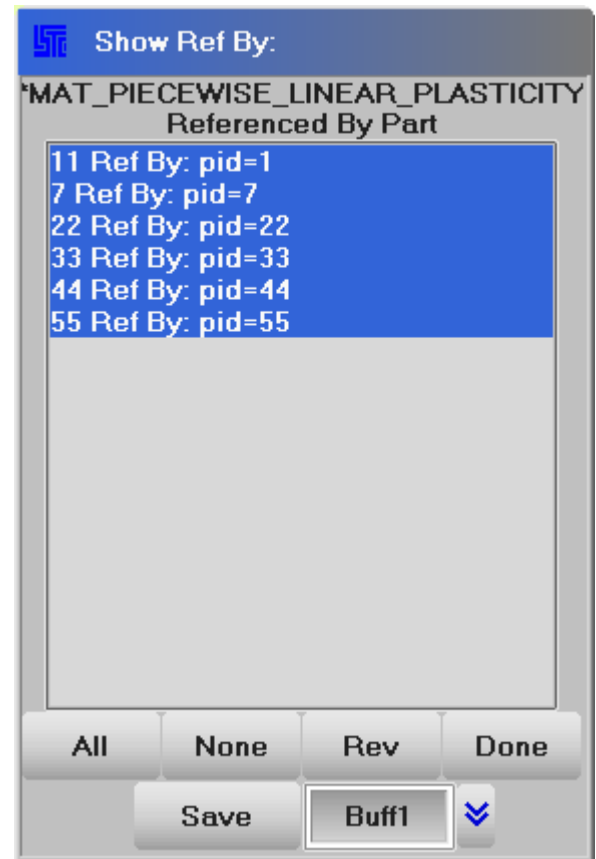




PAGE 3: KEYWORD FILE EDITING

REF BY INTERFACE

- [All] – Select all
- [None] – Deselect all
- [Rev] – Reverse selection
- [Done] – Close Referenced By interface
- [Save] – Save selected part IDs in selected buffer
- [Buff n] – Select buffer to save data





KEYWORD INPUT DIALOG

- [NewID] – Generate a new ID for keyword
- [Clear] – Clear all entries (only on some keyword input dialogs)
- [Pick] – Open pick interface to allow data to be picked directly from the model
- [Add] – Create a fresh keyword using default values
- [Accept] – Accept changes
- [Del/UnD] – Delete/Undelete selected keyword
- [Default] – Set entries to default values
- [Done] – Close Keyword Input dialog, (choose to accept changes)

- Text Field – Enter data(1)
- Link button – Opens Link Dialog(2)
- Choice Box – Select value from list by clicking on the arrow(3)
- Normal Headings – Displays help comment(4)
- Red Headings – Displays help comment, this text field is a dynamic form switch. Entering certain values will unlock further cards for data entry.(5)

Quick Guides

Quick Guide: How to create a keyword using Page 3

1. Open specific keyword form
2. Click Add to open a default form with a new ID (Note each keyword must have a unique internal ID which is generated using Add/NewID. On some forms it may be possible to input a user ID for this purpose.
3. Enter data in specified fields
4. Click Accept to save keyword to memory
5. To save changes permanently the keyword file must be saved via: File/Save keyword



PAGE 3: KEYWORD FILE EDITING

SET DATA

Repeated Data by Button and List

2	N1	N2	N3	N4	A1	A2	A3	A4
					0.0	0.0	0.0	0.0

Range 1 Set Data 2

Data Pt.

Replace

Insert

Delete

Help

[Replace] – Replace selected set data with data in text field

[Insert] – Insert set data after the currently selected one

[Delete] – Delete the selected set data

[Help] – Display input help

Range List – If there are more than 20 set data they are grouped in ranges of 10 set data. This is to allow easy navigation (1)

Set Data List – This is where the previously input set data is displayed (2)

Quick Guides

Quick Guide: Entering Set Data

1. Enter values in associated card
 2. Click [Insert] to add the set data. If set data already exists, the new data will be added after the currently highlighted set.
- Or Click [Replace] to replace the currently highlighted set.

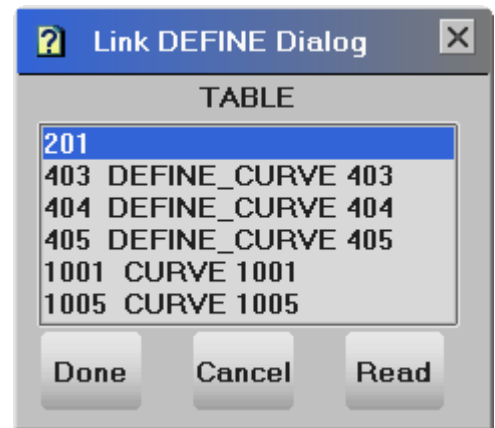


PAGE 3: KEYWORD FILE EDITING

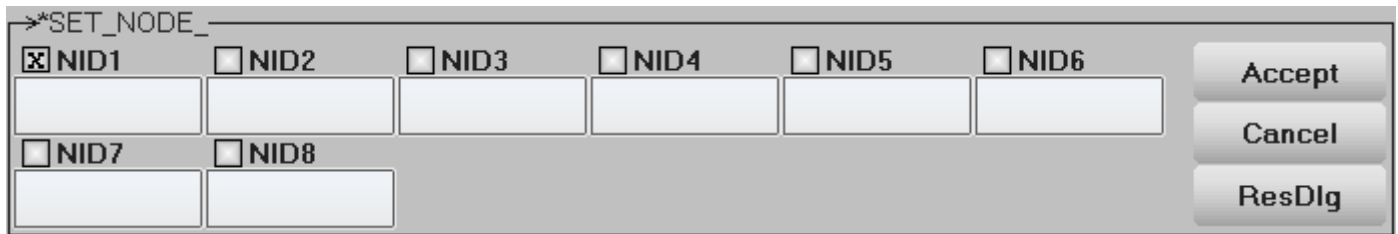
LINK KEYWORD DIALOG

This dialog displays a list of keyword IDs suitable for the text field used.

- [Done] – Accept selection and close Link Keyword Dialog
- [Cancel] – Cancel selection and close Link Keyword Dialog
- [Read] – Open selected keyword (for viewing only)



PICK INTERFACE



- [Accept] – Accept selections and resize Keyword Input dialog
- [Cancel] – Cancel selections and resize Keyword Input dialog
- [ResDlg] – Resize Keyword Input dialog

Quick Guides

Quick Guide: How to pick data from the model

1. Click [Pick] to open Pick Interface
2. Check the field to pick data
3. Pick data by clicking on the model
4. After data has been picked the next field will be checked ready for picking, or check another manually
5. Click [Accept]



PAGE 3: KEYWORD FILE EDITING

DYNAMIC KEYWORD INPUT FORMS

If a keyword input field has a red heading then that field is used as a dynamic form switch. In the following example entering a value greater than one will result in card 2 being replaced. There are various other keywords that have similar abilities and information on the triggering value should be displayed in the help comment for that field.

*MAT_CELLULAR_RUBBER_(TITLE) (0)

TITLE

1

MID	RO	PR	N

2

C10	C01	C11	C20	C02

3

P0	PHI	IVS	G	BETA

N = 0 therefore the standard form is shown.

*MAT_CELLULAR_RUBBER_(TITLE) (0)

TITLE

1

MID	RO	PR	N
			5

2

SGL	SW	ST	LCID

3

P0	PHI	IVS	G	BETA

N > 0 therefore card 2 has changed to allow different keyword data entries.



DEFINE TABLE

KEYWORD INPUT

NewID Pick Add Accept Del/UnD Default Done

*DEFINE_TABLE_(TITLE) (1)

TITLE

1 TBID

201

Repeated Data by Button and List

2

VALUE	LCID
10.00000	403

Data Pt. 1

1	1.0000e+001	403
2	2.0000e+001	404
3	3.0000e+001	405

Replace Insert

Delete Help

Plot Raise

Total Card: 1 Smallest ID: 201 Largest ID: 201 Total deleted card: 0

Define table is a unique keyword form as it allows one to plot data stored in the list.

[Replace] – Replace selected data with data in text fields

[Insert] – Insert data after the currently selected one

[Delete] – Delete the selected data from the list

[Help] – Display input help

[Plot] – Plot load curve data in current XY-Plot window

[Raise] – Raise all open XY-Plot windows

Quick Guides

Quick Guide: How to use define_table

1. Enter a title
2. Enter a table ID
3. Enter a value and a load curve ID
4. Click [Insert]/[Replace] to add to table
5. Click [Plot] to display table data in a XY-Plot window



ENTITY DRAW OPTION

Introduction

If a draw button is shown in the top left of a keyword input form then the keyword entity can be displayed in the drawing area. Once an entity type has been drawn, more of this entity type can be selected using the Entity Operation interface. See instructions under Page 4 section for further details.

Interface Description

[Draw] – Draw entity

[Restore Form]– Restore keyword input form

Quick Guides

Quick Guide: How to display an entity from a keyword input form

1. Select the entity ID from the list on the right side of the form
2. Click [Draw]
(The form will shrink to one side and the selected entity will be displayed)
3. Use the entity operation interface to show/blank multiple entities
4. Click [Restore Form] to retrieve the keyword input form
(The drawn entity will be removed from the drawing area upon restoring the keyword input form)



PAGE 4: PRE PROCESSING TOOLS

ABFold	DmyPos	
IniVel	Accels	DBHist
SpWeld	Spc	Wall
Box	Rivet	GWeld
Coord	Constn	CNRB
SetD	PartD	MassD

INTRODUCTION

The interfaces on this page are all pre processing tools, including airbag folding, dummy manipulation and positioning and entity generation. Although entities can be generated on page 3, the interfaces on this page allow the user to use the drawing area to visualize entities while working on them.



AIRBAG FOLDING

Introduction

Use this interface to define and view air bag folds.

Interface Description

Crea – Create a new fold definition

Mod – Modify an existing fold

Show – Show the selected folds

Fold Definition

ID – Enter fold ID

Title – Enter title for this fold

Fold Type – Select fold type

N1-N2 – Pick/Enter 2 node IDs to define the fold line

Pt+Dir – Define a point and a direction to define the fold line

N1 – Pick/Enter a node ID to define N1

N2 – Pick/Enter a node ID to define N2

N3 – Pick/Enter a node ID to define the active side of the fold

Up – Fold the active side of the fold, up

Down – Fold the active side of the fold, down

Thick – Enter thickness of shell

Scale – Enter scale value

Radius – Enter radius for a circle within which to form the spiral fold

Clearance – Enter clearance value

Ignore – Ignore thickness from previous folds

iMove – Move the fold line nodes back by this amount

Enter the number of steps to take for each fold during animation

Anim – Animate folds upon clicking folding

Loop – Loop animation of folds

[Apply] – Apply changes to fold definition

[Fold] – Execute all folds

[1Fold] – Execute selected fold

[Line] – Create fold lines by cutting elements

[Step] – Step through all folds one at a time

[Reset] – Reset model to original geometry

[Unfold] – Unfold airbag

[Done] – Exit the Airbag Folding interface



```
1 scrunch 114 109 103 +ve Up 1 1 0 Fold_#1
2 scrunch 114 109 103 +ve Up 1 1 0 Fold_#2
3 scrunch 114 109 103 +ve Up 1 1 0 Fold_#3
```

Save

All

Load

None

Cut

Del

[Save] – Save all fold definitions to file

[Load] – Load fold definitions from a file

[Cut] – Cut/paste the selected fold to/from memory

[All] – Select all fold definitions

[None] – Clear fold definition selection

[Delete] – Delete selected fold definitions

Quick Guides

Quick Guide: Creating an airbag fold

1. Click create
2. Enter fold ID and name
3. Select fold type
4. Define fold line by picking 2 nodes
Or by picking 1 node and a direction
5. Pick a node on the active side of the fold. (This active side will be folded up/down/in)
6. Enter fold thickness
7. Enter fold scale for thin/tuck
Or Enter radius for thick/spiral
8. Select direction to fold active side (Up/Down) (Does not apply to tuck/scrunch)
9. Select ignore to ignore previous fold thicknesses
10. Enter iMove value for thin fold only. This will be the distance moved back by the upper (see diagram below)
11. Click [Apply] create fold





Quick Guide: How to simulate all folds

1. Click [Fold] to initiate all defined folds

Quick Guide: How to simulate a single fold

1. Select a fold
2. Click [1Fold]

Quick Guide: Animating folds

1. Enter number of steps to animate per fold
2. Check Anim and click [Step] to move through the folding step by step
3. Check [Loop] to automatically animate all the folds and to keep looping



DUMMY POSITIONING INTERFACE

Introduction

Use this interface to manipulate a HYBRID III dummy.

Interface Description

[Read] – Read in a Tree file

[Write] – Open file write interface

[Reset] – Reset model to last saved settings

[Done] – Exit Dummy Positioning interface.

Sel – Select a rigid body component in current occupant

Cre – Create a rigid body component in current occupant

(Not implemented yet)

Del – Delete selected rigid body component in current occupant

(Not implemented yet)

Positioner interface

H-Point operations – Operations about the H-Point for the entire occupant

Limb operations – Operations affecting joints and limbs

H-point/Global operations

Rotate – Rotate occupant

Translate – Translate occupant

Mirror – Mirror occupant about a plane

H-Point location

X/Y/Z – X/Y/Z coordinates of the H-Point

H-Pt N – Pick/Enter node ID for new H-Point location

→Dummy positioning interface—

Read Write Reset Done

☒ Sel ☐ Cre ☐ Del

→Positioner interface—

☒ H-Point operations

☐ Limb operations

→H-Point/Global operations—

☒ Rotate

☐ Translate

☐ Mirror

H-Point location

X : 0.000000

Y : 0.000000

Z : 0.000000

☐ H-Pt N



PAGE 4: PRE PROCESSING TOOLS

Rotation Interface

The Rotation Interface is a software control panel. It features four radio buttons for axis selection: Global X, Global Z, Global Y (which is selected), and User defined. The 'Rot. Ang.' is set to 10.0000 with a slider below it ranging from -1.0 to +1.0. A 'Div' dropdown is set to 50, and a 'Range' dropdown is set to 1.0. There are checkboxes for 'N1' and 'N2'. An 'Always update' checkbox is checked. At the bottom right are 'Update' and 'Reset' buttons.

Global X/Y/Z – H-Point rotation about global x/y/z axes

User Defined – H-Point rotation about a user defined direction

N1/N2 – Pick/Enter 2 nodes to define a direction

Rot. Ang. – Enter angle to rotate occupant

Div – Select the number divisions used for half the tuning range (i.e. 0 to X or 0 to -X)

Range – Select/Enter angle for the extremes of the fine tuning range

Slider – Drag slider to rotate the occupant about the rotation axis

Always update – Always update occupant's position after operations

[Update] – Update occupant's current position

[Reset] – Revert the last operation performed on the occupant

Translation Interface

The Translation Interface is a software control panel. It features four radio buttons for axis selection: Global X, Global Z, Global Y, and User defined (which is selected). The 'Distance' is set to 0.000000 with a slider below it. There are checkboxes for 'N1' (checked) and 'N2'. An 'Always update' checkbox is checked. At the bottom right are 'Update' and 'Reset' buttons.

Global X/Y/Z – Translate occupant along global x/y/z axes

User Defined – Translate occupant along a user defined direction

N1/N2 – Pick/Enter 2 nodes to define a direction

Distance Factor – Enter distance to translate occupant

Always update – Always update occupant's position after operations

[Update] – Update occupant's current position

[Reset] – Revert the last operation performed on the occupant



Mirror Interface

The Mirror Interface dialog box contains the following elements:

- Four radio buttons for plane selection: **Z(xy) plane** (selected), **Y(zx) plane**, **X(yz) plane**, and **User defined** (indicated by a red circle).
- Three checkboxes for node selection: **N1** (checked), **N2**, and **N3**.
- Three input fields corresponding to the N1, N2, and N3 checkboxes.
- An **Apply** button.
- An **Always update** checkbox (checked).
- Update** and **Reset** buttons.

Z(xy) plane – Mirror occupant with respect to z-plane

X(yz) plane – Mirror occupant with respect to x-plane

Y(zx) plane – Mirror occupant with respect to y-plane

User Defined – Mirror with respect to user defined plane

N1/N2/N3 – Pick/Enter nodes to define a plane

[Apply] – Mirror occupant

Always update – Always update occupant's position after operations

[Update] – Update occupant's current position

[Reset] – Revert the last operation performed on the occupant

Limb Operations interface

Orient Limb – Orient the selected joint

Joint Constraints – Set the joint's orientation constraints

Select limb in the occupant

The Limb Operations interface window contains the following elements:

- A title bar with the text **>Limbs operations**.
- Two radio buttons: **Orient Limb** (selected, indicated by a red circle) and **Joint settings**.
- A list box containing the following items: **lower_torso** (highlighted in blue), **spine**, **upper_torso**, **neck**, **head**, **upper_arm_right**, **lower_arm_right**, **upper_arm_left**, **lower_arm_left**, **upper_leg_right**, **lower_leg_right**, **foot_right**, **upper_leg_left**, **lower_leg_left**, and **foot_left**.



PAGE 4: PRE PROCESSING TOOLS

Limb Orientation interface

Rot. Ang. about N/A

☐ Divs Range

-1.0 1.0

☐ Always update
☐ Lock children
☐ Unlock all

Update
Reset

Rot. Ang. – Enter angle to rotate limb

Div – Select number of divisions for half the fine tuning range

Range – Select/Enter angle for the extremes of the fine tuning range

Slider – Drag slider to rotate the limb about the rotation axis

N/A – ??

Always update – Always update occupant's position after operations

Lock children – Lock children to rotate with current limb

[Update] – Update occupant's current position

[Reset] – Revert the last operation performed on the occupant

Joint setting interface

CP:N

X
Y
Z

☐ Global x-axis ☒ Lock x
☐ Global y-axis ☒ Lock y
☐ Global z-axis ☒ Lock z
☒ User defined ☐ N1

Default

CP:N – Assign control point to another node

X/Y/Z – Enter control point X/Y/Z coordinates

Global X/Y/Z axes – Set x/y/z-axis as the current rotation axis

User Defined – Assign a user defined axis as the current rotation axis

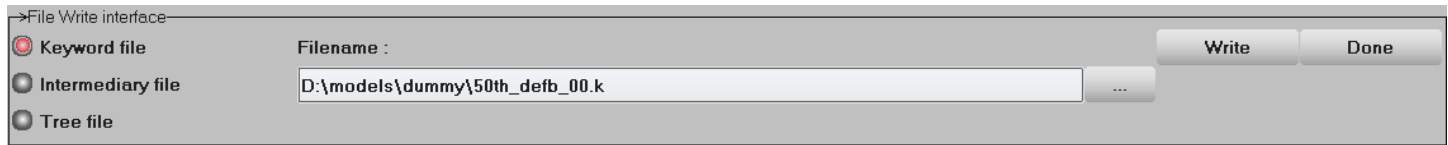
N1– Pick/Enter a node to define a rotation axis. (The CP will be used as the base point)

Lock x/y/z – Prevent the joint from rotating about the x/y/z-axis

[Default] – Reset joint settings to default values



File Write interface



Keyword file – Write data as a keyword file for the occupant model (*.k)

Intermediary file – Write data in keyword format with occupant information (*.inf)

Tree file – Write tree file only

[Filename] – Enter output filename including full path

[...] – Browse to find a directory/filename

[Write] – Write file using selected format

[Done] – Close File Write interface

Quick Guides

Quick Guide: How to load an occupant for dummy positioning

1. File/Open/BinaryPlot
2. Select a .inf file
3. Click [4]
4. Click [DmyPos]
5. Click [Read]
6. Select a tree file for occupant

Quick Guide: How to rotate the occupant about the H-point

1. Select H-Point Operations
 2. Select Rotate
 3. Select rotation axis X,Y,Z
- Or User defined (Pick 2 nodes to form axis of rotation)
4. Enter angle to rotate occupant (Rot. Ang.)
 5. Check Always Update for automatic model update with no undo action (The manual method allows 1 undo action)
 6. Click arrows to rotate occupant
- Or Use fine tuning
- Select number of divisions for half the tuning range (This scale will be applied to full range)
 - Select/Enter angle for the extremes of the fine tuning range
 - Drag slider to adjust fine tuning



Quick Guide: How to translate the occupant

1. Select H-Point Operations
2. Select Translate
3. Select direction of translation X,Y,Z
Or User Defined (Pick 2 nodes to form direction vector)
4. Enter distance to translate occupant
5. Check Always Update for automatic model update with no undo action (The manual method allows 1 undo action)
6. Use arrows to move occupant

Quick Guide: How to mirror the occupant

1. Select H-Point Operations
2. Select Translate
3. Select plane of reflection XY,YZ,XZ
Or User Defined (Pick 3 nodes to form a plane of reflection)
Check Always Update for automatic model update with no undo action (The manual method allows 1 undo action)
4. Click [Apply]

Quick Guide: How to rotate limbs

1. Select Limb operations
2. Select Joint settings
3. Select limb to orientate, either from the list or by right-clicking on the model
4. Lock/Unlock rotation axes
5. Select X/Y/Z axes (if unlocked)
Or pick/enter a node ID to define a rotation axis (The CP is used as the other node)
6. Select Orient limb
7. Check Always Update for automatic model update with no undo action (The manual method allows 1 undo action)
8. Check Lock Children to move children limbs with the parent
9. Left-Click and drag to rotate limb
Or Enter angle of rotation and click arrows to rotate limb
Or Use fine tuning
 - Select number of divisions for half the tuning range (This scale will be applied to full range)
 - Select/Enter angle for the extremes of the fine tuning range
 - Drag slider to adjust fine tuning



PAGE 4: PRE PROCESSING TOOLS

ENTITY OPERATION INTERFACE

The screenshot shows the 'Entity Operation' interface. On the left, there are radio buttons for selection methods: 'Pick Ent', 'Area' (selected), 'Poly', 'In' (selected), 'Out', 'Partial' (selected), and 'Whole'. In the center, there is a dropdown menu for 'Entity Type' set to 'CnstRivet'. Below it, there are radio buttons for operation modes: 'Showing' (selected), 'Blank', 'Ident', and 'KCard'. On the right, there are three buttons: 'Blank All', 'Unblk All', and 'Reverse'.

Introduction

This interface is used with many of the interfaces on Page 4, It allows the user to select entities from the drawing area to display them as well as for modification and deletion.

Interface Description

- Pick Entity – Pick a single entity
- Area – Select entities inside/outside a user defined area
- Polygon – Select entities inside/outside a user defined region
- In – Select entities inside the area/polygon
- Out – Select entities outside the area/polygon
- Partial – Select entities even if only partially within the area
- Whole – Only select entities completely within the area
- Entity Type – Select entity type to be operated upon

Using Show

When Show is selected on the entity interface the following options will be available.

Operation:

- Show – Show entity
- Blank – Blank entity/part/element
- Kcard – Show keyword form for entity

Show All	Show all entities of selected entity type
Show None	Show no entities of selected entity type
Reverse	Reverse selection of entity type

Quick Guides

Quick Guide: How to Show entities on the drawing area

1. Click Show
2. Select selection method (pick/area/polygon, in/out, partial/whole)
3. Clicking on the drawing area will show entities if they exist, it is advisable to use area selection to show entities in a certain region. Use Show all to display all entities.



PAGE 4: PRE PROCESSING TOOLS

Blank All	Blank all entities of selected type
Unblk All	Un-blank all entities of selected type
Reverse	Reverse selection of entity type

Quick Guide: How to Blank entities on the drawing area

1. Click Blank
2. Select entity type, entity/part/element
(Note: Parts / Elements can not be shown once blanked unless [Un Blank All] is used, other entities can be shown using the Show interface)
3. Select entity selection method (pick/area/polygon, in/out, partial/whole)
4. Make selections from the drawing area

Ident all	Identify all entities of selected type
Clear Ident	Clear all identification labels
Reverse	Reverse selection of identified entities

Quick Guide: How to Identify entities on the drawing area

1. Click Identify
2. Select entity selection method (pick/area/polygon, in/out, partial/whole)
3. Select entity type, entity/part/element/node
4. Make selections from the drawing area

Quick Guide: How to use KCard

1. Select KCard
2. Select entity type
3. Pick visible entities to show their keyword data
(Note to pick entities like Box data these must be visible first, so select them from the list or use the Show entity operation)



Using Modify/Delete

Once modify / delete is selected in the entity interface on page 4, the Entity Operations Interface will change. One can no longer show / blank entities. Select is used to highlight/un-highlight displayed entities for modification / deletion. In some cases only a single entity can be modified at a time. In this situation selecting entities using area/polygon selection will only highlight the first entity.

Operation:

Select – Select entities

Kcard – Show keyword form for entity

Select All	Highlight all visible entities
Deselect All	Un-highlight all visible entities
Reverse	Reverse selection

Quick Guides

Quick Guide: How to Select entities

1. Click Select
2. Select entity selection method (pick/area/polygon, in/out, partial/whole)
3. Select entity type
4. Make selections from drawing area

Quick Guide: How to use KCard

This operation is unaffected by the interface change, see above for operation details.



SET DATA

Introduction

Use this interface to show/create/modify/delete Set data.

Interface Description

Show – Show existing Set data

Create – Create new Set data

Modify – Modify existing Set data

Delete – Delete existing Set data

Label – Select label type

[Set_Node, etc] – Select entity type to operate on

Set ID – Enter Set ID. All node sets should have a unique set ID.

Title – Enter title for new/current set (optional).

DA1 – First nodal attribute (default value is 0.0)

DA2 – Second nodal attribute (default value is 0.0)

DA3 – Third nodal attribute (default value is 0.0)

DA4 – Fourth nodal attribute (default value is 0.0)

[All] – Select all sets

[None] – Deselect all sets

[Rev] – Reverse selection

[Apply] – Apply entries for creation/modification

[Cancel] – Cancel entries for modification

[Write] – Write selected sets to a file

[Card] – Open keyword input dialog for this set

[Done] – Exit Set Data interface

SET_SEGMENT extra options

☒ Shell	☐ 2DPart	☐ Outter
☐ Solid	☐ 3DPart	☐ Inner
		☐ all

Shell – Pick shell element to create segment

Solid – Pick solid face to create segment

2DPart – Pick shell part to create segment

3DPart – Pick solid part to create segment

→Set Data

☒ Show ☐ Create
☐ Modify ☐ Delete

Label

None

*SET_NODE

SetID Title(Optional)

DA1 DA2 DA3 DA4

All None Rev Apply

Cancel Write Card Done

70 - NODESET 70
80 - NODESET 80
85 - NODESET 85
90 - NODESET 90



Quick Guides

Quick Guide: Different ways of displaying set data

1. Select Show
2. Select entities from the list to display
3. Select label format
 - None – No label is shown next to entity
 - Symbol – Displays entity ID next to entity

Quick Guide: How to create a Set

1. Select Create
 2. Select set type
 3. Enter unique SetID
 4. Enter optional Title
 5. Enter nodal attributes
 6. Select nodes using General Selection Interface (see above)
 7. Click [Apply] to save changes to memory
- (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify existing Set data

1. Select set data from the list
(Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area. Note these options are only available when Show is selected)
 2. Select Modify (Note: the list and drawing area will only show previously selected entities)
 3. Select a Set from the list
Or Use Select Operation on the Entity Operations Interface to select a Set from the drawing area
 4. Modify nodal attributes
 5. Use General Selection Interface to modify nodes in the set
 6. Click [Apply] to save changes to memory
- (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to delete existing Set data

1. Select entities from the list
(Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area. Note these options are only available when Show is selected)
 2. Select Delete (Note: the list and drawing area will only show previously selected entities)
 3. Select entities from the reduced list
Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
 4. Click [Apply] to delete selected entities
 5. Click [Ok] to confirm deletion
- (Note to save changes permanently use File/Save Keyword)



Quick Guide: How to write out Set data

1. Select Show
2. Select set data
3. Click [Write]
4. Enter filename
5. Click [Save]



*DEFINE_BOX DATA

Introduction

Use this interface to show/create/modify/delete *DEFINE_BOX entities.

Interface Description

Show – Show existing *DEFINE_BOX entities

Create – Create new * DEFINE_BOX entities

Modify – Modify existing * DEFINE_BOX entities

Delete – Delete existing * DEFINE_BOX entities

Label – Select label type

[All] – Select all * DEFINE_BOX entities

[None] – Deselect all * DEFINE_BOX entities

[Rev] – Reverse selection

[AList] – Select all entities within the selected range

[Apply] – Apply entries for creation/modification or selections for deletion

[Cancel] – Cancel entries for modification

[Done] – Exit *Define_Box Data interface

Create

Title – Enter box title

BOXID – Enter box ID

Node1, 2 – Pick minimum and maximum nodes for extremes of box

Areain – Select nodes inside a user defined area for the box

Polyin – Select nodes inside a user defined region for the box

Modify

Title – Enter box title

X – Translate/Scale box in x direction

Y – Translate/Scale box in y direction

Z – Translate/Scale box in z direction

Translate – Translate box

Enter distance to translate box

[Tran-] – Translate in negative direction

[Tran+] – Translate in positive direction

Scale – Scale box

Enter scale factor

[Scale-] – Scale box down

[Scale+] – Scale box up

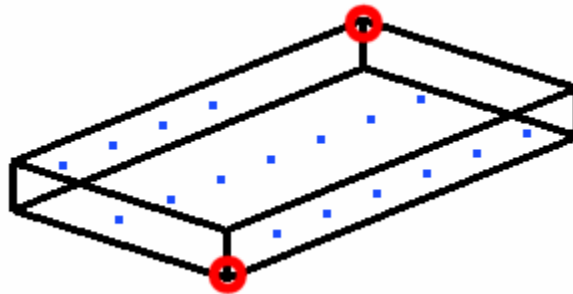
Quick Guides

Quick Guide: Different ways of displaying Box data

1. Select Show
2. Select entities from the list to display
3. Select label format
 - None – No label is shown next to entity
 - Symbol – Displays entity ID next to entity

Quick Guide: How to create a Box

1. Select Create
 2. Select box type
 3. Enter optional Title
 4. Enter box ID
 5. Enter min/max coordinates for extreme vertices
(These can be 2 nodes picked from the model shown in RED)
 - Or Use Area/polygon to pick nodes to be contained by the box (Shown in blue)
 6. Click [Apply] to save changes to memory
- (Note to save changes permanently use File/Save Keyword)



permanently use File/Save



Quick Guide: How to modify existing Box data

1. Select entities from the list
(Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area. Note these options are only available when Show is selected)
2. Select Modify (Note the list and drawing area will only show previously selected entities)
3. Select a box from the list
Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Modify box parameters
5. Use Translate to translate the box
 - Select Translate
 - Select direction
 - Enter distance using model units
 - Click Trans +/- to translate box
6. Use Scale to change the dimensions of the box.
 - Select Scale
 - Select direction
 - Enter scale factor
 - Click Scale +/- to scale box
7. Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to delete existing Box data

1. Select entities from the list
(Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area. Note these options are only available when Show is selected)
2. Select Delete (Note: the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
4. Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
5. Click [Apply] to delete selected entities
6. Click [Ok] to confirm deletion
(Note to save changes permanently use File/Save Keyword)



*DEFINE_COORDINATE DATA

Introduction

Use this interface to show/create/modify/delete *DEFINE_COORDINATE entities.

Interface Description

Interface Description

Show – Show existing * DEFINE_COORDINATE entities

Create – Create new * DEFINE_COORDINATE entities

Modify – Modify existing * DEFINE_COORDINATE entities

Delete – Delete existing * DEFINE_COORDINATE entities

Label – Select label type

Coord Type – Select coordinate type

CID – Coordinate ID. A unique number has to be defined

[New ID] – Generate a new ID for keyword

Title – Enter a title for the coordinate entity

[All] – Select all * DEFINE_COORDINATE entities

[None] – Deselect all * DEFINE_COORDINATE entities

[Rev] – Reverse selection

[AList] – Select all entities within the selected range

[Apply] – Apply entries for creation/modification or selections for deletion

[Cancel] – Cancel entries for modification

[Done] – Exit * Define_Coordinate Data interface

Coordinate Type Options

*Nodes

N1 – Enter node ID for node at local origin

N2 – Enter node ID for node on local x-axis

N3 – Enter node ID for node on local x-y plane

Flag – Set to unity, 1, if the local system is to be updated each time step for the BOUNDARY_SPC nodal constraints and ELEMENT_BEAM type 6, the discrete beam element. Generally, this option when used with nodal SPC's is not recommended since it can cause excursions in the energy balance because the constraint forces at the node may go through a displacement if the node is partially constrained

*System

XO – x-coordinate of origin

YO – y-coordinate of origin

ZO – z-coordinate of origin

XL – x-coordinate of point on local x-axis

YL – y-coordinate of point on local x-axis



ZL – z-coordinate of point on local x-axis
XP – x-coordinate of point in local x-y plane
YP – y-coordinate of point in local x-y plane
ZP – z-coordinate of point in local x-y plane

***Vector**

XX – x-coordinate on local x-axis. Origin lies at (0,0,0)
YX – y-coordinate on local x-axis
ZX – z-coordinate on local x-axis
XV – x-coordinate of local x-y vector
YV – y-coordinate of local x-y vector
ZV – z-coordinate of local x-y vector

Quick Guides

Quick Guide: Different ways of displaying Coordinate data

1. Select Show
2. Select entities from the list to display
3. Select label format
 - None – No label is shown next to entity
 - Symbol – Displays entity ID next to entity

Quick Guide: How to create Coordinate data

1. Select Create
2. Select coordinate type
3. Enter unique Coordinate System ID
Or Click [NewID]
4. Enter Title
5. Enter coordinate type options
(Note: Use check boxes to pick nodes from the drawing area)
6. Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify existing Coordinate data

1. Select entities from the list
(Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area.
Note these options are only available when Show is selected)
2. Select Modify (Note: the list and drawing area will only show previously selected entities)
3. Select an entity from the list
Or Use Select Operation on the Entity Operations Interface to select one from the drawing area
4. Modify parameters
5. Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)



Quick Guide: How to delete existing Coordinate data

1. Select entities from the list
(Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area.
Note these options are only available when Show is selected)
2. Select Delete (Note: the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Click [Apply] to delete selected entities
5. Click [Ok] to confirm deletion
(Note to save changes permanently use File/Save Keyword)



PAGE 4: PRE PROCESSING TOOLS

*CONSTRAINED_GENERALIZED_WELD_SPOT DATA (GeWELD)

Introduction

Use this interface to show/create/modify/delete *CONSTRAINED_GENERALIZED_WELD_SPOT entities.

Interface Description

Show – Show existing *CONSTRAINED_GENERALIZED_WELD_SPOT entities

Create – Create new *CONSTRAINED_GENERALIZED_WELD_SPOT entities

Modify – Modify existing

*CONSTRAINED_GENERALIZED_WELD_SPOT entities

Delete – Delete existing *CONSTRAINED_GENERALIZED_WELD_SPOT entities

Label – Select label type

[New ID] – Generate a new ID for keyword

NSID – Enter Node Set ID

[KCard] / [Entitylist] – Toggle between keyword card entry interface and entity list

[All] – Select all *CONSTRAINED_GENERALIZED_WELD_SPOT entities

[None] – Deselect all *CONSTRAINED_GENERALIZED_WELD_SPOT entities

[Rev] – Reverse selection

[AList] – Select all entities within the selected range (Only applies when there are more than 20 entities in the list)

[Apply] – Apply entries for creation/modification or selections for deletion

[Done] – Exit Constrained Generalized Weld Spot Data interface



Quick Guides

Quick Guide: Different ways of displaying Constrained Generalized Weld Spot data

1. Select Show
2. Select entities from the list to display
3. Select label format
 - None – No label is shown next to entity
 - Symbol – Displays entity ID next to entity

Quick Guide: How to create a Constrained Generalized Weld Spot

1. Select Create
 2. Enter NSID, or click [NewID] to generate a new one
 3. Select nodes using General Selection Interface
 4. Click [EditParam] to modify optional keyword data
 5. Click [Apply] to save changes to memory
- (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify existing Constrained Generalized Weld Spot data

1. Select entities from the list
(Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area. Note these options are only available when Show is selected)
 2. Select Modify (Note the list and drawing area will only show previously selected entities)
 3. Select Sngl / Multi
Select Single Modification Mode – Use to modify entity nodes and parameters. When an entity is selected from the drawing area the modification interface will open automatically.
(Note: When selecting from the drawing area, once modification mode is active, entity selection is not possible)
 - Modify parameters and click [Apply] to save changes to memory
 - Or click [Cancel] and select another entity
- Or Select Multiple Modification Mode – Use to modify parameters of multiple entities. When an entity is selected it is only highlighted, so additional entities can be selected without modifying the first. Once all selections have been made click [Edit]
- Modify parameters and Click [Apply] to save changes to memory
 - Or click [Cancel]

(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to delete existing Constrained Generalized Weld Spot data

1. Select entities from the list
(Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area. Note these options are only available when Show is selected)
 2. Select Delete (Note: the list and drawing area will only show previously selected entities)
 3. Select entities from the reduced list
- Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Click [Apply] to delete selected entities
 5. Click [Ok] to confirm deletion
- (Note to save changes permanently use File/Save Keyword)



*CONSTRAINED_SPOT_WELD DATA (SPWELD)

Introduction

Use this interface to show/create/modify/delete

*CONSTRAINED_SPOTWELD entities.

Interface Description

Show – Show existing *CONSTRAINED_SPOT_WELD entities

Create – Create new *CONSTRAINED_SPOT_WELD entities

Modify – Modify existing *CONSTRAINED_SPOT_WELD entities

Delete – Delete existing *CONSTRAINED_SPOT_WELD entities

Label – Select label type

Auto Create – Auto create spot weld

N1,N2 – Pick/Enter Node ID 1 and 2

Show Parameters – Toggle view spot weld data parameters

[All] – Select all *CONSTRAINED_SPOTWELD entities

[None] – Deselect all *CONSTRAINED_SPOTWELD entities

[Rev] – Reverse selection

[AList] – Select all *CONSTRAINED_SPOTWELD entities within the selected range (Only applies when there are more than 20 entities in the list)

[Apply] – Apply entries for creation/modification or selections for deletion

[Cancel] – Cancel entries for modification

[Done] – Exit Spot Weld Data interface

Quick Guides

Quick Guide: Different ways of displaying Constrained Spot Weld data

1. Select Show
2. Select entities from the list to display
3. Select label format
 - None – No label is shown next to entity
 - Symbol – Displays entity ID next to entity
 - Detail – Displays entity ID and SpWeld node IDs next to entity

The screenshot shows the 'Spot Weld Data' dialog box. At the top, there are four radio buttons: 'Show' (selected), 'Create', 'Modify', and 'Delete'. Below these is a 'Label' dropdown menu currently set to 'None'. A 'ShowPara' checkbox is also present. Underneath, there are labels 'WID', 'N1' (with a pick icon), and 'N2' (with a checkbox). Below these are three empty input fields. A row of four buttons follows: 'All', 'None', 'Rev', and 'AList'. Below that is a row of three buttons: 'Apply', 'Cancel', and 'Done'. The bottom half of the dialog is a large, empty list area for displaying the selected entities.



PAGE 4: PRE PROCESSING TOOLS

Quick Guide: How to create a Constrained Spot Weld

(See keyword user's manual section on *CONSTRAINED_SPOT_WELD for parameter details)

Entity SpotWData Parameter					
SN	SS	N	M	TF	EP
				1.0E+20	1.0E+20

1. Select Create
 2. Enter parameters
 3. Autocreate enabled (Enables creation without having to click apply after every entry)
 - Pick/Enter 2 node IDs
 4. Or Autocreate disabled
 - Pick/Enter 2 node IDs
 5. Click [Apply] to save changes to memory
- (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify existing Constrained Spot Weld data

1. Select entities to modify from the list
(Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area. Note these options are only available when Show is selected)
 2. Select Modify (Note the list and drawing area will only show previously selected entities)
 3. Select Sngl / Multi
Select Single Modification Mode – Use to modify SpWeld nodes and parameters. When an entity is selected from the list or drawing area the modification interface will open automatically.
(Note: When selecting from the drawing area, once modification mode is active, entity selection is not possible)
 - Modify parameters and click [Apply] to save changes to memory
 - Or click [Cancel]
- Or Select Multiple Modification Mode – Use to modify parameters of multiple entities. When an entity is selected from the drawing area it is only highlighted, so another entity can be selected without modifying the first. Once all selections have been made click [Edit]
- Modify parameters and Click [Apply] to save changes to memory
 - Or click [Cancel]
- (Note to save changes permanently use File/Save Keyword)



*CONSTRAINED_SPOT_WELD DATA (SPWELD)

Introduction

Use this interface to show/create/modify/delete

*CONSTRAINED_SPOTWELD entities.

Interface Description

Show – Show existing *CONSTRAINED_SPOT_WELD entities

Create – Create new *CONSTRAINED_SPOT_WELD entities

Modify – Modify existing *CONSTRAINED_SPOT_WELD entities

Delete – Delete existing *CONSTRAINED_SPOT_WELD entities

Label – Select label type

Auto Create – Auto create spot weld

N1,N2 – Pick/Enter Node ID 1 and 2

Show Parameters – Toggle view spot weld data parameters

[All] – Select all *CONSTRAINED_SPOTWELD entities

[None] – Deselect all *CONSTRAINED_SPOTWELD entities

[Rev] – Reverse selection

[AList] – Select all *CONSTRAINED_SPOTWELD entities within the selected range (Only applies when there are more than 20 entities in the list)

[Apply] – Apply entries for creation/modification or selections for deletion

[Cancel] – Cancel entries for modification

[Done] – Exit Spot Weld Data interface

Quick Guides

Quick Guide: Different ways of displaying Constrained Spot Weld data

4. Select Show
5. Select entities from the list to display
6. Select label format
 - None – No label is shown next to entity
 - Symbol – Displays entity ID next to entity
 - Detail – Displays entity ID and SpWeld node IDs next to entity



PAGE 4: PRE PROCESSING TOOLS

Quick Guide: How to create a Constrained Spot Weld

(See keyword user's manual section on *CONSTRAINED_SPOT_WELD for parameter details)

Entity SpotWData Parameter					
SN	SS	N	M	TF	EP
				1.0E+20	1.0E+20

6. Select Create
 7. Enter parameters
 8. Autocreate enabled (Enables creation without having to click apply after every entry)
 - Pick/Enter 2 node IDs
 9. Or Autocreate disabled
 - Pick/Enter 2 node IDs
 10. Click [Apply] to save changes to memory
- (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify existing Constrained Spot Weld data

4. Select entities to modify from the list
(Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area. Note these options are only available when Show is selected)
 5. Select Modify (Note the list and drawing area will only show previously selected entities)
 6. Select Sngl / Multi
Select Single Modification Mode – Use to modify SpWeld nodes and parameters. When an entity is selected from the list or drawing area the modification interface will open automatically.
(Note: When selecting from the drawing area, once modification mode is active, entity selection is not possible)
 - Modify parameters and click [Apply] to save changes to memory
 - Or click [Cancel]
- Or Select Multiple Modification Mode – Use to modify parameters of multiple entities. When an entity is selected from the drawing area it is only highlighted, so another entity can be selected without modifying the first. Once all selections have been made click [Edit]
- Modify parameters and Click [Apply] to save changes to memory
 - Or click [Cancel]
- (Note to save changes permanently use File/Save Keyword)



Quick Guide: How to delete existing Constrained Spot Weld data

1. Select entities from the list
(Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area.
Note these options are only available when Show is selected)
2. Select Delete (Note the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Click [Apply] to delete selected entities
5. Click [Ok] to confirm deletion
(Note to save changes permanently use File/Save Keyword)



*CONSTRAINED_SPOT_WELD DATA (SPWELD)

Introduction

Use this interface to show/create/modify/delete

*CONSTRAINED_SPOTWELD entities.

Interface Description

Show – Show existing *CONSTRAINED_SPOT_WELD entities

Create – Create new *CONSTRAINED_SPOT_WELD entities

Modify – Modify existing *CONSTRAINED_SPOT_WELD entities

Delete – Delete existing *CONSTRAINED_SPOT_WELD entities

Label – Select label type

Auto Create – Auto create spot weld

N1,N2 – Pick/Enter Node ID 1 and 2

Show Parameters – Toggle view spot weld data parameters

[All] – Select all *CONSTRAINED_SPOTWELD entities

[None] – Deselect all *CONSTRAINED_SPOTWELD entities

[Rev] – Reverse selection

[AList] – Select all *CONSTRAINED_SPOTWELD entities within the selected range (Only applies when there are more than 20 entities in the list)

[Apply] – Apply entries for creation/modification or selections for deletion

[Cancel] – Cancel entries for modification

[Done] – Exit Spot Weld Data interface

Quick Guides

Quick Guide: Different ways of displaying Constrained Spot Weld data

7. Select Show
8. Select entities from the list to display
9. Select label format
 - None – No label is shown next to entity
 - Symbol – Displays entity ID next to entity
 - Detail – Displays entity ID and SpWeld node IDs next to entity



PAGE 4: PRE PROCESSING TOOLS

Quick Guide: How to create a Constrained Spot Weld

(see keyword user's manual section on *CONSTRAINED_SPOT_WELD for parameter details)

Entity SpotWData Parameter					
SN	SS	N	M	TF	EP
				1.0E+20	1.0E+20

11. Select Create
 12. Enter parameters
 13. Autocreate enabled (Enables creation without having to click apply after every entry)
 - Pick/Enter 2 node IDs
 14. Or Autocreate disabled
 - Pick/Enter 2 node IDs
 15. Click [Apply] to save changes to memory
- (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify existing Constrained Spot Weld data

7. Select entities to modify from the list
(Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area.
Note these options are only available when Show is selected)
 8. Select Modify (Note the list and drawing area will only show previously selected entities)
 9. Select Sngl / Multi
Select Single Modification Mode – Use to modify SpWeld nodes and parameters. When an entity is selected from the list or drawing area the modification interface will open automatically.
(Note: When selecting from the drawing area, once modification mode is active, entity selection is not possible)
 - Modify parameters and click [Apply] to save changes to memory
 - Or click [Cancel]
- Or Select Multiple Modification Mode – Use to modify parameters of multiple entities. When an entity is selected from the drawing area it is only highlighted, so another entity can be selected without modifying the first. Once all selections have been made click [Edit]
- Modify parameters and Click [Apply] to save changes to memory
 - Or click [Cancel]
- (Note to save changes permanently use File/Save Keyword)



*CONSTRAINED_NODAL_RIGID_BODY DATA (CNRB)

Introduction

Use this interface to show/create/modify/delete *CONSTRAINED_NODAL_RIGID_BODY entities.

Interface Description

Show – Show existing *CONSTRAINED_NODAL_RIGID_BODY entities

Create – Create new *CONSTRAINED_NODAL_RIGID_BODY entities

Modify – Modify existing *CONSTRAINED_NODAL_RIGID_BODY entities

Delete – Delete existing *CONSTRAINED_NODAL_RIGID_BODY entities

Label – Select label type

[New ID] – Generate a new ID for keyword

PID – Enter Part ID of the nodal rigid body

CID – Enter Coordinate ID/Open link dialog by clicking on the label

NSID – Enter Node Set ID

Pick PNODE – Pick node from model to be PNODE

PNODE – Enter PNODE/Open link dialog by clicking on the label

IPRT – Select print flag

[All] – Select all *CONSTRAINED_NODAL_RIGID_BODY entities

[None] – Deselect all *CONSTRAINED_NODAL_RIGID_BODY entities

[Rev] – Reverse selection

[AList] – Select all entities within the selected range

[Apply] – Apply entries for creation/modification or selections for deletion

[Done] – Exit Constrained Nodal Rigid Body Data interface

Quick Guides

Quick Guide: Different ways of displaying CNRB data

1. Select Show
2. Select entities from the list to display
3. Select label format
 - None – No label is shown next to entity
 - Symbol – Displays entity ID next to entity

Quick Guide: How to create a CNRB

1. Select Create
2. Enter parameters
3. Enter a unique NSID or click New ID to generate a new one
4. Select nodes using General Selection Interface
5. Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)



Quick Guide: How to modify existing CNRB data

1. Select entities from the list
(Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area. Note these options are only available when Show is selected)
2. Select Modify (Note the list and drawing area will only show previously selected entities)
3. Select a single entity from the list
Or Use Select Operation on the Entity Operations Interface to select an entity from the drawing area
4. Modify parameters
5. Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to delete existing CNRB data

1. Select entities from the list
(Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area. Note these options are only available when Show is selected)
2. Select Delete (Note the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Click [Apply] to delete selected entities
5. Click [Ok] to confirm deletion
(Note to save changes permanently use File/Save Keyword)



*INITIAL_VELOCITY DATA (INIVEL)

Introduction

Use this interface to show/create/modify/delete *INITIAL_VELOCITY entities.

Interface Description

Show – Show existing *INITIAL_VELOCITY entities
Create – Create new *INITIAL_VELOCITY entities
Modify – Modify existing *INITIAL_VELOCITY entities
Delete – Delete existing *INITIAL_VELOCITY entities

Label – Select label type

Vx/Vy/Vz – Enter initial velocity in x/y/z directions

Vrx/Vry/Vrz – Enter initial rotational velocity in x/y/z directions

[All] – Select all *INITIAL_VELOCITY entities

[None] – Deselect all *INITIAL_VELOCITY entities

[Rev] – Reverse selection

[AList] – Select all entities within the selected range

[Apply] – Apply entries for creation/modification or selections for deletion

[Done] – Exit Initial Velocity Data interface

[Done] – Exit Constrained Nodal Rigid Body Data interface

Quick Guides

Quick Guide: Different ways of displaying Initial Velocity data

1. Select Show
2. Select entities from the list to display
3. Select label format
 - None – No label is shown next to entity
 - Symbol – Displays entity ID next to entity
 - Detail – Displays entity ID and linear velocity components next to entity

Quick Guide: How to set an Initial Velocity

1. Select Create
2. Enter linear velocity in X/Y/Z directions
3. Enter rotational velocity in X/Y/Z directions
4. Select nodes using General Selection Interface
5. Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)

Initial Velocity Data

☒ Show ☐ Create

☐ Modify ☐ Delete

Label None

Vx

Vy

Vz

Vrx

Vry

Vr

All None Rev

AList Apply Done



Quick Guide: How to modify existing Initial Velocity data

1. Select entities from the list
(Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area.
Note these options are only available when Show is selected)
2. Select Modify (Note the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Modify parameters
5. Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to delete existing Initial Velocity data

1. Select entities from the list
(Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area.
Note these options are only available when Show is selected)
2. Select Delete (Note the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Click [Apply] to delete selected entities
5. Click [Ok] to confirm deletion
(Note to save changes permanently use File/Save Keyword)



SPC DATA

Introduction

Use this interface to show/create/modify/delete SPC data.

Interface Description

Show – Show existing SPC nodes

Create – Create new SPC nodes

Modify – Modify existing SPC nodes

Delete – Delete existing SPC nodes

Label – Select label type

X,Y,Z – Toggle translational constraint in local X,Y,Z directions respectively

RX,RY,RZ – Toggle rotational constraint about the local X,Y,Z axes respectively

CID – Open Define_Coordinate_{Option} link dialog

[All] – Select all SPC nodes

[None] – Deselect all SPC nodes

[Rev] – Reverse selection

[AList] – Select all entities within the selected range

[Apply] – Apply entries for creation/modification or selections for deletion

[Done] – Exit SPC Data interface

Quick Guides

Quick Guide: Different ways of displaying SPC data

1. Select Show
2. Select entities from the list to display
3. Select label format
 - None – No label is shown next to entity
 - Symbol – Displays entity ID next to entity
 - Detail – Displays entity ID and the number of the active constraints next to entity

Quick Guide: How to create an SPC keyword

1. Select Create
2. Set translational constraint directions
3. Set rotational constraint directions
4. Select/Enter CID value
5. Select nodes using General Selection Interface
6. Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)

The SPC Data dialog box is a vertical window with a title bar labeled "SPC Data". It contains several sections: a top section with four radio buttons labeled "Show", "Create", "Modify", and "Delete"; a section with a "Label" dropdown menu currently set to "Symbol"; a section with six checkboxes labeled "X", "Y", "Z", "RX", "RY", and "RZ"; a section with a "CID" label and an adjacent text input field; a section with three buttons labeled "All", "None", and "Rev"; a section with three buttons labeled "AList", "Apply", and "Cancel"; and a large empty rectangular area at the bottom for a list of entities.



Quick Guide: How to modify SPC data

1. Select SPC data from the list
(Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area.
Note these options are only available when Show is selected)
2. Select Modify (Note the list and drawing area will only show previously selected entities)
3. Select SPC data from the list
Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Modify translational constraint directions (X, Y, Z)
5. Modify rotational constraint directions (RX, RY, RZ)
6. Modify CID value
7. Modify node selection using General Selection Interface
8. Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to delete existing SPC data

1. Select entities from the list
(Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area.
Note these options are only available when Show is selected)
2. Select Delete (Note the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Click [Apply] to delete selected entities
5. Click [Ok] to confirm deletion
(Note to save changes permanently use File/Save Keyword)



CONSTRAINED RIVET DATA (RIVET)

Introduction

Use this interface to show/create/modify/delete Constrained Rivet Data.

Interface Description

Show – Show existing *CONSTRAINED_RIVET entities

Create – Create new *CONSTRAINED_RIVET entities

Modify – Modify existing *CONSTRAINED_RIVET entities

Delete – Delete existing *CONSTRAINED_RIVET entities

Label – Select label type

RID – Enter Rivet ID (Optional)

N1 – Pick/Enter node ID 1

N2 – Pick/Enter node ID 2

TF – Enter failure time for nodal constraint set

[All] – Select all *CONSTRAINED_RIVET entities

[None] – Deselect all *CONSTRAINED_RIVET entities

[Rev] – Reverse selection

[AList] – Select all entities within the selected range

[Apply] – Apply entries for creation/modification or selections for deletion

[Done] – Exit Initial Velocity Data interface

[Done] – Exit Constrained Rivet Data interface

Quick Guides

Quick Guide: Different ways of displaying Constrained Rivet data

1. Select Show
2. Select entities from the list to display
3. Select label format
 - None – No label is shown next to entity
 - Symbol – Displays entity ID next to entity
 - Detail – Displays entity ID and CR node IDs next to entity

The screenshot shows the 'Rivet Data' dialog box. At the top, there are four radio buttons: 'Show' (selected), 'Create', 'Modify', and 'Delete'. Below these are three input fields: 'Label' with a dropdown menu showing 'Ion', a small blue arrow button, and a checkbox for 'Autocreate'. The 'RID' field has a checkbox for 'N1' (checked) and a checkbox for 'N2'. Below these are three input boxes for 'N1', 'N2', and 'TF'. At the bottom, there are four buttons: 'All', 'None', 'Rev', and 'AList'. Below these are three buttons: 'Apply', 'Cancel', and 'Done'. The main area of the dialog is a large empty box for displaying the data.



Quick Guide: How to create a Constrained Rivet

1. Select Create
 2. Enter RID and TF
 3. Autocreate enabled (Enables creation without having to click apply after every entry)
 - Pick/Enter 2 node IDs
 - Or Autocreate disabled
 - Pick/Enter 2 node IDs
 - Click [Apply] to save changes to memory
- (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify existing Constrained Rivet data

1. Select entities from the list
(Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area. Note these options are only available when Show is selected)
 2. Select Modify (Note the list and drawing area will only show previously selected entities)
 3. Select a single entity from the list
Or Use Select Operation on the Entity Operations Interface to select one from the drawing area
 4. Modify RID, TF and CR nodes
 5. Autocreate enabled (Enables creation without having to click apply after every entry)
 - Pick/Enter 2 node IDs
 - Or Autocreate disabled
 - Pick/Enter 2 node IDs
 - Click [Apply] to save changes to memory
- (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to delete existing Constrained Rivet data

1. Select entities from the list
(Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area. Note these options are only available when Show is selected)
 2. Select Delete (Note the list and drawing area will only show previously selected entities)
 3. Select entities from the reduced list
Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
 4. Click [Apply] to delete selected entities
 5. Click [Ok] to confirm deletion
- (Note to save changes permanently use File/Save Keyword)



CONSTRAINED NODE SET DATA (CONSTN)

Introduction

Use this interface to show/create/modify/delete Constrained Node Set Data.

Interface Description

Show – Show existing *CONSTRAINED_NODE_SET entities

Create – Create new *CONSTRAINED_NODE_SET entities

Modify – Modify existing * CONSTRAINED_NODE_SET entities

Delete – Delete existing * CONSTRAINED_NODE_SET entities

Label – Select label type

RID – Enter Rivet ID (Optional)

N1 – Pick/Enter node ID 1

N2 – Pick/Enter node ID 2

TF – Enter failure time for nodal constraint set

[All] – Select all * CONSTRAINED_NODE_SET entities

[None] – Deselect all * CONSTRAINED_NODE_SET entities

[Rev] – Reverse selection

[AList] – Select all entities within the selected range

[Apply] – Apply entries for creation/modification or selections for deletion

[Done] – Exit Constrained Node Set Data interface

Quick Guides

Quick Guide: Different ways of displaying Constrained Node Set data

1. Select Show
2. Select entities from the list to display
3. Select label format
 - None – No label is shown next to entity
 - Symbol – Displays entity ID next to entity

>CONSTN Data

☒ Show ☐ Create

☐ Modify ☐ Delete

Label **None** ▼

CNSID NSID

New ID

DOF **1** ▼ TF **1.0E+20**

All None Rev AList

Apply Cancel Done



Quick Guide: How to create a Constrained Node Set

1. Select Create
2. Enter Constrained Node Set ID and Node Set ID
Or Click [New ID]
3. Select nodes using General Selection Interface
4. Select DOF value
5. Enter TF value
6. Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify existing Constrained Node Set data

1. Select entities from the list
(Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area.
Note these options are only available when Show is selected)
2. Select Modify (Note the list and drawing area will only show previously selected entities)
3. Select a single entity from the list
4. Or Use Select Operation on the Entity Operations Interface to select one from the drawing area
5. Modify parameters
6. Select nodes using General Selection Interface
7. Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to delete existing Constrained Node Set data

1. Select entities from the list
(Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area.
Note these options are only available when Show is selected)
2. Select Delete (Note the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Click [Apply] to delete selected entities
5. Click [Ok] to confirm deletion
(Note to save changes permanently use File/Save Keyword)



DATABASE HISTORY DATA

Introduction

Use this interface to assign database history nodes/beams/ shells/solids/sph nodes/tshells.

Interface Description

Show – Show existing *DATABASE_HISTORY_{OPTION} entities

Create – Create new *DATABASE_HISTORY_{OPTION} entities

Modify – Modify existing *DATABASE_HISTORY_{OPTION} entities

Delete – Delete existing *DATABASE_HISTORY_{OPTION} entities

Type – Select database history data type

Label – Select label type

[All] – Select all *DATABASE_HISTORY_{OPTION} entities

[None] – Deselect all *DATABASE_HISTORY_{OPTION} entities

[Rev] – Reverse selection

[AList] – Select all entities within the selected range

[Apply] – Apply entries for creation/modification or selections for deletion

[Done] – Exit Database History Data interface

Quick Guides

Quick Guide: Different ways of displaying Database History data

1. Select Show
2. Select entities from the list to display
3. Select label format
 - None – No label is shown next to entity
 - Symbol – Displays entity ID next to entity



Quick Guide: How to create Database History data

1. Select Create
2. Enter Constrained Node Set ID and Node Set ID
Or Click [New ID]
3. Select nodes using General Selection Interface
4. Select DOF value
5. Enter TF value
6. Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify existing Database History data

1. Select entities from the list
(Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area.)
2. Note these options are only available when Show is selected)
3. Select Modify (Note the list and drawing area will only show previously selected entities)
4. Select/Deselect nodes using General Selection Interface
(Note: existing entities may also be selected / deselected)
5. Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to delete existing Database History data

1. Select entities from the list
(Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area.
Note these options are only available when Show is selected)
2. Select Delete (Note the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Click [Apply] to delete selected entities
5. Click [Ok] to confirm deletion
(Note to save changes permanently use File/Save Keyword)



MASS DATA

Introduction

Use this interface to show/create/modify/delete Mass data.

Interface Description

Show – Show existing mass data

Create – Create new mass data

Modify – Modify existing mass data

Delete – Delete existing mass data

Label – Select label type

Mass Value – Enter mass value

[All] – Select all mass nodes

[None] – Deselect all mass nodes

[Rev] – Reverse selection

[AList] – Select all entities within the selected range

[Apply] – Apply entries for creation/modification or selections for deletion

[Done] – Exit Mass Data interface

Mass Data

☒ Show ☐ Create

☐ Modify ☐ Delete

☐ Label

Mass Value

Quick Guides

Quick Guide: How to create Mass nodes

Select Create

Enter mass value

Select nodes using General Selection Interface

Click [Apply] to save changes to memory

(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify Mass nodes

Select an SPC keyword from the list

Or use Selection on Entity Operations Interface

(This interface is only shown in Show/Delete modes)

Select Modify

Modify mass value

Modify node selection using General Selection Interface

Click [Apply] to save changes to memory

(Note to save changes permanently use File/Save Keyword)



PART DATA

Introduction

Use this interface to show/create/modify/delete Part data.

Interface Description

Show – Show existing part data

Create – Create new part data

Modify – Modify existing part data

Sear – Search for parts using user defined parameters

Assi – Assign entered data to selected parts

Part Type – Select Part Entity type

Pick – Pick a single part

Area – Select entities inside/outside a user defined area

Poly – Select entities inside/outside a user defined region

In – Select entities inside the area/polygon

Out – Select entities outside the area/polygon

[All] – Select all parts

[None] – Deselect all parts

[Rev] – Reverse selection

[Del] – Delete selected parts

[Write] – Write selected parts to a file

[Done] – Exit Part Data interface

The screenshot shows a software interface titled "Part Data". It features several sections of controls:

- Top Section:** Three radio buttons labeled "Show", "Crea", and "Modi". The "Show" button is selected.
- Second Section:** Two radio buttons labeled "Sear" and "Assi".
- Third Section:** A label "Part Type" followed by a dropdown menu currently showing "Beam" with a blue arrow icon.
- Fourth Section:** Three radio buttons labeled "Pick", "Area", and "Poly". The "Pick" button is selected.
- Fifth Section:** Two radio buttons labeled "In" and "Out".
- Sixth Section:** A grid of six buttons: "All", "None", "Rev", "Del", "Write", and "Done".
- Bottom Section:** A large, empty rectangular area for displaying data.



PAGE 4: PRE PROCESSING TOOLS

Part ID		
New ID	Add	Accept

[New ID] – Generate a new ID for keyword

[Add] - Generate a new ID for keyword and set all fields to default values

[Accept] - Accept entries for current part

Part Title							
SECID	MID	EOSID	HGID	TMID	GRAV	0	▼
					ADPOPT	0	▼

PID – Part ID

SECID – Enter Section ID /Open link dialog by clicking on the label

MID – Enter Material ID /Open link dialog by clicking on the label

EOSID – Enter Equation of state ID /Open link dialog by clicking on the label

HGID – Enter Hourglass/bulk viscosity ID /Open link dialog by clicking on the label

GRAV – Select part initialization for gravity loading

ADPOPT – Select part adaptivity

TMID – Enter Thermal material property ID

Search

SECID	MID	EOSID	HGID	TMID	GRAV	ADPOPT
					NA	NA
Load RefBy					Buff1	▼
					Search Part	Apply

[Load RefBy] – Load Part IDs stored in selected reference buffer

[Buffer n] – Select buffer to load Part IDs

Assign

Assign Part	Apply
--	--------------------------------------



Quick Guides

Quick Guide: How to create Parts

1. Select Create
 2. Enter Part ID
- Or Click [New ID]/[Add]
3. Select part type
 4. Enter keyword data
 5. Click [Apply] to save changes to memory

(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify Parts

1. Select a part from the list
- Or use Selection on Entity Operations Interface
(This interface is only shown in Show/Delete modes)
2. Select Modify
 3. Modify Part ID
 4. Modify part type
 5. Modify keyword data
 6. Click [Apply] to save changes to memory

(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to use Search

1. Enter values in fields to search for parts containing those IDs
 2. Click [Apply]
- Or
1. Select Buffer
 2. Click Load Ref By – Material/EOS/Hourglass keywords that are used in Part keywords can be saved into buffers using the appropriate dialog window on Page 3. (See Page3 instructions) These buffers can be reloaded here to display the parts.

Quick Guide: How to assign new keyword values to multiple parts

1. Select parts from list / model
2. Enter new keyword data
3. Click [Apply] to assign new values

(Note to save changes permanently use File/Save Keyword)



PAGE D: DISPLAY INTERFACE

INTRODUCTION

Use this interface to display entities that are present in the current keyword file.

INTERFACE DESCRIPTION

[All] – Turn on all entities

[None] – Turn off all entities

Select entity type to be displayed

[All] – Turn on all entities in the current list

[None] – Turn off all entities in the current list

[Rev] – Reverse selection

[AList] – Turn on all entities in the selected range only

Label – Select label type. Note: not all entities have a detailed label, in this situation selecting Detail will only display a symbol style label.



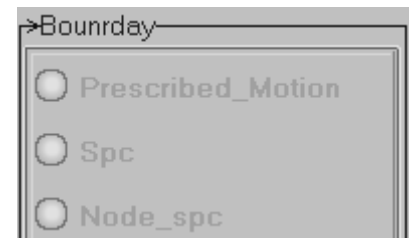
The Entity Operation interface can be used, see description under Page 4 chapter.

BOUNDARY

Prescribed motion – Display *BOUNDARY_PRESCRIBED_{OPTION} data

SPC – Display *BOUNDARY_SPC_{OPTION} data

Node_SPC – Display nodal boundary SPC data (See *NODE_NODE data)



CONSTRAINED

ExNodes – Display *CONSTRAINED_EXTRA_NODES_{OPTION} data

GenWeld – Display *CONSTRAINED_GENERALIZED_WELD_SPOT data

SpotWeld – Display *CONSTRAINED_SPOT_WELD data

RgBodies – Display *CONSTRAINED_RIGID_BODY_{OPTION} data

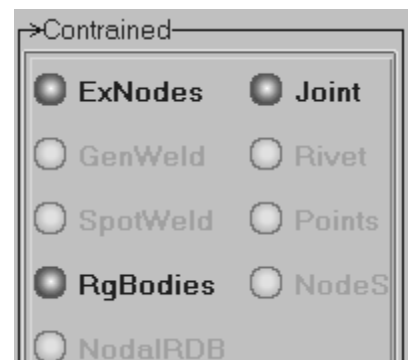
NodalRDB – Display *CONSTRAINED_NODAL_RIGID_BODY data

Joint – Display *CONSTRAINED_JOINT_{OPTION} data

Rivet – Display *CONSTRAINED_RIVET data

Points – Display *CONSTRAINED_POINTS data

NodeS – Display *CONSTRAINED_NODE_SET data





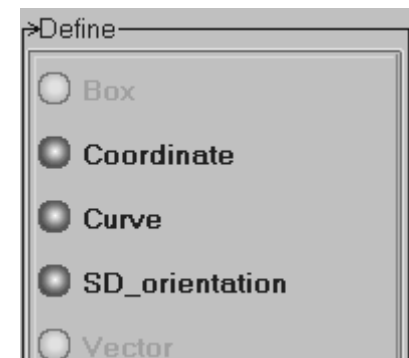
CONTACT

General – Display *CONTACT_GENERAL_{OPTION} data
Entity – Display *CONTACT_ENTITY data
Gebod – Display *CONTACT_GEBOD_{OPTION} data
Interior – Display *CONTACT_INTERIOR data
RigidSurface – Display *CONTACT_RIGID_SURFACE data
1D – Display *CONTACT_1D_{OPTION} data
2D – Display *CONTACT_2D_{OPTION} data



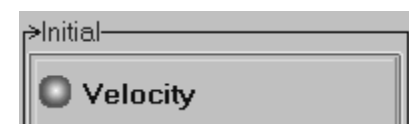
DEFINE

Box – Display *DEFINE_BOX_{OPTION} data
Coordinate – Display *DEFINE_COORDINATE_{OPTION} data
Curve – Display *DEFINE_CURVE_{OPTION} data
SD_orientation – Display *DEFINE_SD_ORIENTATION data
Vector – Display *DEFINE_VECTOR data



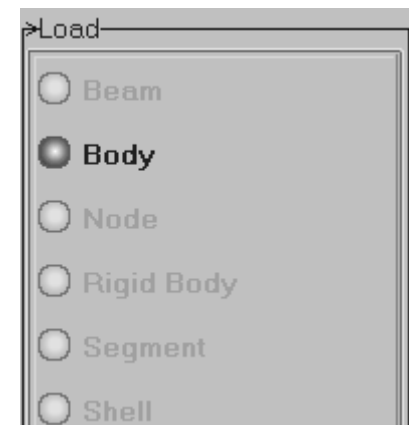
INITIAL

Velocity – Display *INITIAL_VELOCITY_{OPTION} data



LOAD

Beam – Display *LOAD_BEAM_{OPTION} data
Body – Display *LOAD_BODY_{OPTION} data
Node – Display *LOAD_NODE_{OPTION} data
Rigid Body – Display *LOAD_RIGID_BODY data
Segment – Display *LOAD_SEGMENT_{OPTION} data
Shell – Display *LOAD_SHELL_{OPTION} data





SET

Beam – Display *SET_BEAM_{OPTION} data
Discrete – Display *SET_DISCRETE_{OPTION} data
Node – Display *SET_NODE_{OPTION} data
Part – Display *SET_PART_{OPTION} data
Segment – Display *SET_SEGMENT_{OPTION} data
Shell – Display *SET_SHELL_{OPTION} data
Solid – Display *SET_SOLID_{OPTION} data
Tshell – Display *SET_TSHELL_{OPTION} data
SPH – Display *SET_SPH data

>Set

☐ Beam ☐ Shell

☐ Discrete ☐ Solid

☒ Node ☐ Tshell

☒ Part ☐ Sph

☒ Segment

SEAT BELT

Accelerometer – Display *ELEMENT_SEATBELT_ACCELEROMETER data
Pretensioner – Display *ELEMENT_SEATBELT_PRETENSIONER data
Reactor – Display *ELEMENT_SEATBELT_REACTOR data
Sensor – Display *ELEMENT_SEATBELT_SENSOR data
Slipring – Display *ELEMENT_SEATBELT_SLIPRING data

>SeatB

☒ Accelerometer

☐ Pretensioner

☐ Retractor

☐ Sensor

☐ Slipring

RIGID WALL

Rigidwall – Display *RIGID_WALL_{OPTION} data

>Rigidwall

☒ Rigidwall

QUICK GUIDES

Quick Guide: Displaying entities using Page D

1. Select main entity type (using choice box)
2. Select sub entity group from the top list
3. Select members of that sub-entity group from the bottom list to display
(See Entity Interface for instructions regarding the lower interface)



APPENDIX I

CONFIGURATION FILE OPTIONS

Introduction

The .lspostrc file stores information regarding the LS-Pre/Post configuration. This file is only present if the user saves a configuration that is different from the defaults within LS-Pre/Post.

LS-Pre/Post will look for this file in the following locations:

1. LSTC_FILE – An environment variable defining the directory where the LSTC license key resides
2. \$HOME – User home directory
3. WD – working directory where the model resides

LS-Pre/Post searches through these three locations in order, options set in location 1 will be overwritten if found in location 2 and overwritten again if found in location 3. Note this is at the option level and so if different options are set in different locations LS-Pre/Post will use them all instead of defaults. Any options not found in user defined .lspostrc files will be set to default automatically by Ls-Pre/Post.

List of Configuration File Options

max_physical_memory = 1867392K – assign maximum available physical memory for LS-Pre/Post

texture = off – Switch texture mode off/on (only applicable when in shade mode)

twoside_lighting = on – Switch between Two-sided (on) /One-sided (off) lighting

background_color = 0.000, 0.000, 0.000 – Set indicated color using RGB range (0.0 to 1.0)

text_color = 1.000, 1.000, 1.000 – Set indicated color using RGB range (0.0 to 1.0)

mesh_color = 0.000, 0.000, 0.000 – Set indicated color using RGB range (0.0 to 1.0)

label_color = 1.000, 1.000, 1.000 – Set indicated color using RGB range (0.0 to 1.0)

hilite_color = 0.500, 0.500, 0.500 – Set indicated color using RGB range (0.0 to 1.0)

outline_color = 0.000, 0.000, 0.000 – Set indicated color using RGB range (0.0 to 1.0)

triad_color = 1.000, 1.000, 1.000 – Set indicated color using RGB range (0.0 to 1.0)

edgewidth = 1 – Set edge view line width

meshlwidth = 1 – Set mesh view line width

fringe_light = on – Switch fringe component lighting on/off

plotunode = off – Switch plot un-referenced nodes on/off

plotmode = shade – Switch model plot mode shade/view/wire/feat/edge/grid

outlining = edge – Switch outlining type edge/feat/off

shade_mode = flat – Switch shading mode flat/smooth

highlight_node = off – Assign default setting for highlighting nodes under Node Trace, on/off

result_onscreen = off – Switch results on screen off/on (Only applicable if a model entity is identified with show results toggled on)

highlight = on – Switch highlight on/off

mat_ambient = 0.200000 – Assign default value for material light ambience (0.0 to 1.0)



mat_diffuse = 0.200000 – Assign default value for material light diffusion (0.0 to 1.0)
mat_specular = 0.700000 – Assign default value for material light speculance (0.0 to 1.0)
mat_shininess = 50.000000 – Assign default value for material shininess (0.0 to 128.0)
light_ambient.1 = 0.300, 0.300, 0.300 – Assign ambient light RGB values from (0.0 to 1.0)
light_diffuse.1 = 0.700, 0.700, 0.700 – Assign diffuse light RGB values from (0.0 to 1.0)
light_specular.1 = 0.400, 0.400, 0.400 – Assign specular light RGB values from (0.0 to 1.0)
light_position.1 = 0.000, 0.000, 1.000 – Assign light 1 xyz co-ordinates
transparency_value = 0.850 – Assign default transparency value (0.0 = opaque to 1.0 = transparent)
mat_color.1 = 0.769, 0.004, 0.110 – Assign up to 29 different material colors for the color map
fringe_level = 10 – Set default number of fringe levels to display 2 to 30
vect_plotscale = 1.000 – Assign default value for vector arrow scale factor
sph_spherescale = 0.300 – Assign default value for sph sphere scale factor
sph_spheredivs = 12 – Assign default value for number of segments to draw an sph sphere
sph_spherestyle = 4 – Assign default value for sph sphere style,
(1 = dot, 2 = flat, 3 = wire frame polygon, 4 = shiny sphere)
sph_fixedradius = 0 – Assign default radius for sph sphere
shrink_factor = 0.800 – Assign default value for shrink scale factor
rotate_angle = 10.0 – Assign default value for model rotation
feature_angle = 30.0 – Assign angle limit for feature line generation
default_light = 0.300, 0.700, 0.400 – Assign default light color with RGB values from (0.0 to 1.0)
help_label = off – Switch help bubbles off/on
mainmenu_on = right – Set position of main menu right/left
mainwindow_x = 1423 – Set default window size
mainwindow_y = 1068 – Set default window size
plotwindow_x = 0 – Set XY-Plot window start x-coordinate
plotwindow_y = 0 – Set XY-Plot window start y-coordinate
xyplotlegend = on – Switch default setting for XY-Plot window legend on/off
init_path = c:\dynafiles\ – Assign default directory for files to be read (Note: directory must exist)
session_file = lspost.cfile – Assign file name for command file
message_file = lspost.msg – Assign file name for message file
message = off – Switch default setting for message recording off/on
ic_gravity = 9.81 – Assign the value for gravity
ic_timeunit = 1 – Assign the value for standard time unit
drawscene = on – Assign default model drawing option, (on = model drawn upon opening file,
off = model is not drawn upon opening file)
shell_ipt = maxima – Assign default position of shell surface integration points maxima/upper/middle/lower
beam_tubesegs = 8 – Assign number of segments used to draw a beam as a tube
beam_prismon = off – Display beam as prism on/off
beam_sizefactor = 500 – Assign beam prism scale factor
ge_local_brickstrain = 0 – Assign default value for green strain
animate_mode = 0 – Animate mode (0 = loop animation, 1 = animate once)
duplicate_action = 2 – Assign default action when duplicated data is found when reading a keyword file
(0 = ignore, 1 = replace all, 2 = ask user for action)



The following menu page options show current interface button assignments, these can be changed here, however it is easier to use the Assign Menu Button dialog within LS-Pre/Post. (One can assign up to 7 pages):

menu_page1 = 1,2,3,4,5,6,7,8,9,10,11,12,13,14,15,16,17,18,19,20,21,22,23,24,25,26,27

polygon_offset_factor = 1 – Assign default polygon outline offset factor

polygon_offset_unit = 1 – Assign default polygon outline offset unit

override_screen_aspect = 0 – Assign default window aspect ratio

fld_engstrain = 1 – Assign default value for FLD engineering strain

fld_tvalue = 0.8 – Assign default value for FLD t

fld_nvalue = 0.21 – Assign default value for FLD n

def_limit FLC = 0 – Assign default value for FLD FLC limit

def_allow_thinning = 0.3 – Assign default value for FLD allowable thinning

def_safety_margin = 20 – Assign default value for FLD safety margin

def_essential_thinning = 0.02 – Assign default value for FLD required thinning

def_allow_thickening = 0.01 – Assign default value for FLD allowable thickening

def_r_value = 1.82 – Assign default value for FLD ‘r’

def_failure = on – Assign default value for FLD failure on/off

def_risk_fracture = on – Assign default value for FLD risk of fracture on/off

def_severe_thinning = on – Assign default value for FLD severe thinning on/off

def_inadequate_stretch = on – Assign default value for FLD lack of stretching on/off

def_wrinkle_tendency = on – Assign default value for FLD wrinkle tendency on/off

def_wrinkles = on – Assign default value for FLD wrinkles on/off

* use a backslash at end of line to continue points list on next line

* _____10_____20_____30_____40_____50_____60_____70_____80

def FLC = DEF 800521501

-0.3 0.909, -0.2 0.666, -0.1 0.475, 0 0.357, 0.1 0.44, \

0.2 0.493, 0.3 0.523, 0.4 0.547, – Define FLC with paired values (separate individual values with a space, separate pairs by comas)

pscompression_mode = none – Assign default postscript compression mode none/gz/irle/lzw

entitylabel_size = 4 – Assign default entity label size

primitive_size = 0.6 – Assign default primitive size factor

thickenedshell = 0 – Assign default setting for drawing a shell with thickness as a solid (0 = off, 1 = on)